BRITISH STANDARD 308 : 1953

ENGINEERING DRAWING PRACTICE

BRITISH STANDARDS INSTITUTION

F.W. JONES.

BRITISH STANDARD
ENGINEERING DRAWING PRACTICE

B.S. 308 : 1953

Price 10/6 net

BRITISH STANDARDS INSTITUTION

INCORPORATED BY ROYAL CHARTER

BRITISH STANDARDS HOUSE, 2 PARK ST., LONDON, W.1

TELEGRAMS : STANDARDS, AUDLEY, LONDON   TELEPHONE : MAYFAIR 9000

THIS BRITISH STANDARD, having been approved by the Mechanical Engineering Industry Standards Committee and endorsed by the Chairman of the Engineering Divisional Council, was published under the authority of the General Council on 30th December, 1953.

First published September, 1927
First revision December, 1943
Second revision December, 1953.

The Institution desires to call attention to the fact that this British Standard does not purport to include all the necessary provisions of a contract.

In order to keep abreast of progress in the industries concerned, British Standards are subject to periodical review. Suggestions for improvements will be recorded and in due course brought to the notice of the committees charged with the revision of the standards to which they refer.

A complete list of British Standards, numbering over 2000, indexed and cross-indexed for reference, together with an abstract of each standard, will be found in the Institution's Yearbook, price 12s. 6d.

This standard makes reference to the following British Standards :—
B.S. 499 : Section 7. Scheme of symbols for welding.
B.S. 1134. The assessment of surface texture.
B.S. 1265. Engineers' pattern drawing boards.
B.S. 1936. Undercuts and run outs for screw threads.

British Standards are revised, when necessary, by the issue either of amendment slips or of revised editions. It is important that users of British Standards should ascertain that they are in possession of the latest amendments or editions.

CO-OPERATING ORGANIZATIONS

The Mechanical Engineering Industry Standards Committee, under whose supervision this British Standard was prepared, consists of representatives from the following Government departments and scientific and industrial organizations :—

*Admiralty
Air Ministry
Associated Office Technical Committee
Association of Consulting Engineers' Incorporated
British Chemical Plant Manufacturers' Association
British Compressed Air Society
*British Electrical and Allied Manufacturers' Association
British Electricity Authority and Area Boards
British Engineers' Association
British Internal Combustion Engine Manufacturers' Association
British Iron and Steel Executive
*British Railways, The British Transport Commission
Crown Agents for the Colonies
D.S.I.R.—Mechanical Engineering Research Laboratory
Engineering Equipment Users' Association
Gas Council
High Commission of India
Institute of Marine Engineers
Institute of Petroleum
*Institution of Civil Engineers
Institution of Gas Engineers
Institution of Heating and Ventilating Engineers
*Institution of Mechanical Engineers
Institution of Mechanical Engineers (Automobile Division)
*Institution of Production Engineers
Locomotive Manufacturers' Association
Machine Tool Trades' Association
Ministry of Fuel and Power
Ministry of Labour and National Service, Factory Department
*Ministry of Supply
Ministry of Transport
*Ministry of Works
*National Physical Laboratory
*Radio Industry Council
War Office

The Government departments and scientific and industrial organizations marked with an asterisk in the above list, together with the following, were directly represented on the committee entrusted with the preparation of this standard :—

Association of Engineering and Shipbuilding Draughtsmen
Drawing Office Material Manufacturers' and Dealers' Association
H.M. Stationery Office
Imperial College of Science and Technology
Institution of Engineering Draughtsmen and Designers
Ministry of Education
Society of British Aircraft Constructors
Society of Motor Manufacturers and Traders

MEE/10

CONTENTS

BRITISH STANDARD
ENGINEERING DRAWING PRACTICE

FOREWORD

This British Standard, which has been prepared under the authority of the Mechanical Engineering Industry Standards Committee, supersedes B.S. 308 : 1943, 'Engineering drawing office practice'. The most important changes concern the section on Dimensioning and Tolerancing, which has been greatly amplified in the light of valuable experience gathered during the intervening years. In other respects, the section on General Practice has been confined to recommended principles and methods to be followed in the preparation of engineering drawings. Subjects such as architectural drawings, survey plans, graphs, drawing reproduction, and the nature and handling of drawing materials have been excluded, as it is considered that such matters are more fully and appropriately dealt with in other British Standards. The appendix on British Standards for materials specifications and general engineering, which included a list of British Standards for rolled steel sections and pipes, has also been omitted, on the ground that such references may become out of date in course of time, and can always be obtained quite readily and in more detail from other B.S.I. publications.

In view of the above changes in the character of the standard, it has been considered appropriate to alter the title from 'Engineering drawing office practice' to 'Engineering drawing practice'.

In its approach to the section on Dimensioning and Tolerancing, the technical committee responsible for drafting this standard came to the conclusion that there was indisputable evidence that in some circumstances customary dimensioning and tolerancing, however well applied, did not necessarily ensure that components would assemble or function correctly. This section has therefore been drafted, firstly to ensure a reasonably uniform practice in the expression of drawing dimensions and tolerances, and secondly to provide means of tolerancing such geometrical characteristics as straightness, flatness, parallelism, squareness, angularity, concentricity, symmetry, and position.

It should be clearly understood, however, that it is not suggested that the whole comprehensive system of tolerancing need be applied unless the circumstances indicate that material advantages will accrue from the increased drawing work entailed. It rests with the design organization to decide how far it is necessary to specify geometrical tolerances in each particular instance, having regard to functional requirements, interchangeability and probable manufacturing circumstances. Drawings prepared for widespread mass production at home or abroad, or for sub-contracting in workshops of widely varying equipment and experience, may be quoted as particular cases in which the most complete and explicit tolerancing is desirable, and in fact necessary. In such circumstances, reference back to the designer by those responsible for production and inspection is often quite impracticable, and the information given on the drawing must be so complete in dimensional and geometric requirements as to enable the work to be made, and inspected, to suit the full requirements of the design. On the other hand, a large proportion of work can be produced quite satisfactorily without resorting to tolerances on geometrical form, or even without the necessity for tolerancing the bulk of dimensions which need not be machined to very fine limits. This is particularly true where products are specially made in small quantities in a self-contained works, in which manufacturing technique, machine tool equipment and inspection organization have been built up through years of specialized experience. It is considered, however, that when the principles of geometrical tolerancing become more widely known, an increasing number of cases will be found where the full or partial application of the more complete tolerancing system set out in this standard will enable the design requirements to be expressed more exactly than hitherto, with advantage to the design, manufacturing and inspection departments.

This foreword would not be complete without an acknowledgement of the pioneer work of the Inter-Services Committee on Dimensioning and Tolerancing, which led to the publication, in 1948, by H.M. Stationery Office of 'Dimensional Analysis of Engineering Designs'. The fundamental principles established in that publication form the basis on which the Dimensioning and Tolerancing section of this revised British Standard have been considered, and those who wish to make a deeper study of the subject will find 'Dimensional Analysis of Engineering Designs' a useful volume of reference.

NOTE. It is worthy of mention that a Conference on Drawing Practice was held in Ottawa in November, 1950, between Service representatives of U.S.A., Canada and the United Kingdom, with the collaboration of industrial delegates from Canada and Britain. The British Delegation, which consisted mainly of members of the committee responsible for drafting this revision of B.S. 308, tabled a preliminary draft of the proposed section on Dimensioning and Tolerancing, which was discussed in full detail. It is significant to record that it was unanimously agreed by the Conference that drawings produced in this country in accordance with the principles of this standard would be more clearly understood in Canada and U.S.A. than hitherto. Standards of a similar nature submitted by the U.S.A. Munitions Board Standards Agency were also discussed in detail at the Conference, and a very real measure of uniformity was attained by mutual adjustments of the proposals tabled by the three countries.

A further American—British—Canadian (A.B.C.) Conference

was held in New York in June, 1952, at which both Services and Industrial interests of all three countries were well represented ; and discussions are continuing. It is expected that these will result in a common understanding of the principles involved and in the unification of the methods by which those principles should be expressed on drawings. It is evident, therefore, that if the drawing offices in the three countries follow the practices recommended in their respective standards, the mutual interpretation of drawings will be materially facilitated.

Contacts have also been made during 1951, 1952 and 1953 with various European countries, through the International Organization for Standardization (I.S.O.) and further I.S.O. Conferences will be held in the future. It is hoped that the I.S.O. Standards to be issued in due course will embody many of the principles and practices on which this standard is based.

NOTES ON READING THE STANDARD

A. The figures used in the text of this standard are complete only in so far as is necessary to illustrate the point in question and they are not intended to be fully dimensioned working drawings. Except where otherwise stated the dimensions shown in the figures are in inches.

B. Similarly, the numerical values of dimensions and tolerances given in the standard are typical only and are not quoted as recommended practices.

C. The tolerance diagrams given in some of the figures to illustrate the interpretation of the various tolerances are provided for explanatory purposes only and are not intended to appear on the component drawing.

D. In the illustrations to this standard, capital letters are used for notes which are intended to appear on the finished drawing; lower case letters are used for explanatory notes which should **not** appear on the drawing.

E. Clauses 1 to 10 of the standard deal with general drawing practice and in these clauses the illustrations have been executed using first angle projection.

Clauses 11 to 23 cover the subjects of dimensioning and tolerancing and, in these, third angle projection has been employed. The plates representing typical working drawings, given at the end of the standard, illustrate the use of both methods of projection.

F. In general, the terms used in this standard are in conformity with the definitions given in B.S. ......, 'Definitions for use in mechanical engineering'.*

* In course of preparation.

SECTION ONE : GENERAL PRACTICE

1. SIZES OF DRAWINGS AND TRACINGS

a. *Selection of sizes.* A list of standard sizes of drawing sheets in common use is given in Column 1, Table 1. From this list a number of sizes—marked with an asterisk—have been selected as preferred sizes on the basis of the following consideration :

The economical use of standard rolls of drawing and tracing paper, tracing cloth and sensitized paper and cloth, all of which are manufactured in nominal 30 in. and 40 in. widths. When cut or printed sheets are used, these in the first place have to be cut from 30 in. and 40 in. rolls.

It is recognised that many sizes other than those marked as preferred have been in general use for a long time, and that drawing boards, plan cabinets, and filing receptacles have been designed to accommodate them. In such cases, for reasons of economy, it may be necessary to continue the use of non-preferred sizes.

It is however recommended that the preferred sizes be adopted wherever it is economically possible to do so.

TABLE 1. STANDARD SIZES

Nominal sizes of drawings and tracings. Overall dimensions in inches across edges of sheet. (See Fig. 1)	Maximum frame size. Dimensions in inches between border lines (See Fig. 1)
A B	C D
*40 × 72	*38 × 70
*40 × 60	*38 × 58
*30 × 53	*29 × 52
*30 × 40	*29 × 39
27 × 40	26 × 39
15 × 40	14 × 39
22 × 30	21 × 29
*20 × 30	*19 × 29
20 × 27	19 × 26
*15 × 20	*14 × 19
*10 × 15	* 9¼ × 14¼
†13 × 8	12¼ × 7¼
†10 × 8	9¼ × 7¼

* Preferred sizes.

† Additional sizes to be in conformity with commercial stationery. Drawings which are to be filed in binders and which have heights of 13 in. or 10 in. may have widths greater than 8 in.

It is to be noted that provision has been made for a blank border which will enable prints to be trimmed if necessary without incurring the use of uneconomical sizes of sensitized material to allow for trimming. Finished prints which have been trimmed will then be slightly less in size than the original drawings but not less than the frame or border line size of the drawings.

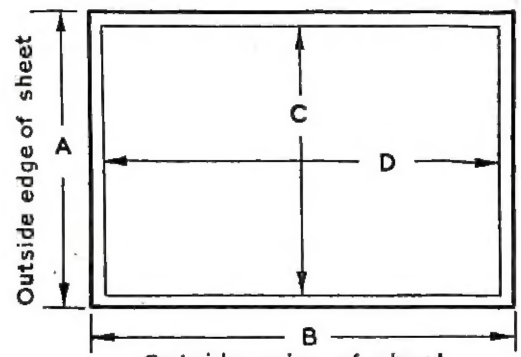


Fig. 1. Drawing sheet sizes

b. *General.* (i) Dimensions C and D (Fig. 1) are the recommended maximum border line sizes; these border lines may be drawn further in from the edge of the sheet if desired, for example, to provide a binding margin, or an additional allowance for trimming.

(ii) Drawing, tracing and sensitized sheets should be cut to sizes A × B. After printing, the sensitized sheets may be trimmed if necessary but it is recommended that the prints be left as large as possible. The border lines which appear on the drawings and tracings should not be trimmed off the prints.

(iii) No notes or figures which are required to be reproduced on the print should appear outside the border lines.

(iv) On the commercial stationery size sheets, border lines may be omitted if desired, but it is recommended that the drawing space utilized should not exceed that determined by the C and D dimensions.

(v) It is recommended that all drawings and tracings should be marked with the standard sheet size (A × B) in inches, or an identification letter, in a convenient place.

NOTE. For information on British Standard drawing boards reference should be made to B.S. 1265, ' Engineers' pattern drawing boards '.

2. LAYOUT OF DRAWINGS

a. *Title blocks.* It is recognized that latitude may be necessary in the arrangement and position of title blocks, etc. to meet differing requirements, but it is considered that the adoption by any particular organization of a standard layout of sheet will facilitate the reading of drawings and make it possible for essential references to be located easily. Many drawing offices use sheets of standard size and layout which are bought already cut to size and printed.

In general, the title block is located at the bottom right hand corner of the sheet, but for convenience of drawing layout the top right hand corner may sometimes be preferred.

It is recommended that spaces should be provided in all title blocks for the following information :—

Name of firm.
Drawing number.
Description or title of drawing.
Scale.
Date and signature.

b. *Repetition of drawing number.* The drawing number may be repeated in other corners of the drawing to ensure that it is visible when the drawing is filed.

c. *Additional information.* Where appropriate, the layout of the standard sheet may include or provide for the following additional information and notes :

(i) Material or parts list. When several parts are detailed on the same sheet, it is usual to tabulate them in a material or parts list, which should preferably be placed adjacent to the title block (see Figs. 3, 4 and 5). Alternatively, this list may appear on a separate sheet (see Clause 2 *e*). In either case it should include such information as :—

Item or part number.
Description of part.
Quantity required.
Cross-references to other detail drawings (if the parts are not actually detailed on the new drawing).
Material (where appropriate, this should be clearly specified to be in accordance with a British Standard or other accepted specification, and the exact grade of the particular material defined as may be necessary).

(ii) Job or order number.

(iii) Treatment, finish, etc.

(iv) Key to machining and other symbols.

(v) Tolerances on dimensions not individually toleranced.

(vi) Tool and gauge references.

(vii) General notes. (For examples see Clause 15, page 43, in Section Two.)

In addition to the foregoing, individual drawing offices will naturally add to their drawings other information suited to their special requirements.

d. *Typical title blocks.* Typical examples incorporating the above recommendations are illustrated in Figs. 2, 3, 4, 5 and 6.

Figure 2. This is suitable for detail drawings, and particularly where one part only is shown on a given sheet, the size of which will vary to suit the actual detail. For convenience of manufacture it is preferable to include all necessary information on the drawing, and the title block should therefore cover such information as :—

Name of firm.	Material form and treatment.
Drawing or part number.	Finish.
Name of part.	Key to symbols.
Scale.	General tolerances for dimensions not individually toleranced.
Material specification.	Date and signature.

Figures 3 *and* 4. These are suitable for assembly drawings and for detail drawings which show a number of parts on the same sheet. In either case, much of the necessary information will be given in a material or parts list (see Clause 2 *c* (i)) and the title block need only provide for the general information specified in Clause 2 *a*.

Figure 5. This is suitable for drawings of shop equipment, such as jigs, tools and gauges, where quantities are not usually involved, and it is often useful to include details and assembly on one sheet. The title block may cover such information as :—

Name of firm.	Scale.
Drawing number.	Key to symbols.
Job number.	General tolerances.
Description.	Date and signature.

Figure 6. This layout is a typical case where the frame size of the drawing has been reduced to suit folding to particular dimensions and to provide a margin for binding.

e. *Separate material or parts lists.* It is sometimes found convenient to group a number of associated parts on a separate sheet distinct from the detail drawing or drawings. Such a list may be applied to a number of parts which are all detailed on one large drawing ; alternatively it may be used to group a number of single-part drawings, or even several multi-part drawings, into an assembly or sub-assembly. A typical form of separate material or parts list is shown in Fig. 7.

NOTE. Some organizations use an independent series of item numbers (e.g. 1, 2, 3, 4, etc.) to identify the parts comprising the assembly ; others find it convenient to identify the parts by their detail drawing numbers. It is also sometimes necessary to allocate other part reference numbers for special purposes, such as spare parts catalogues. These independent reference numbers can conveniently be indicated on the separate parts list.

f. *Drawing lists.* It is frequently necessary to compile a list of all drawings, assembly lists or sub-assembly lists comprising a complete project. A typical example of a separate sheet for this purpose is shown in Fig. 8.

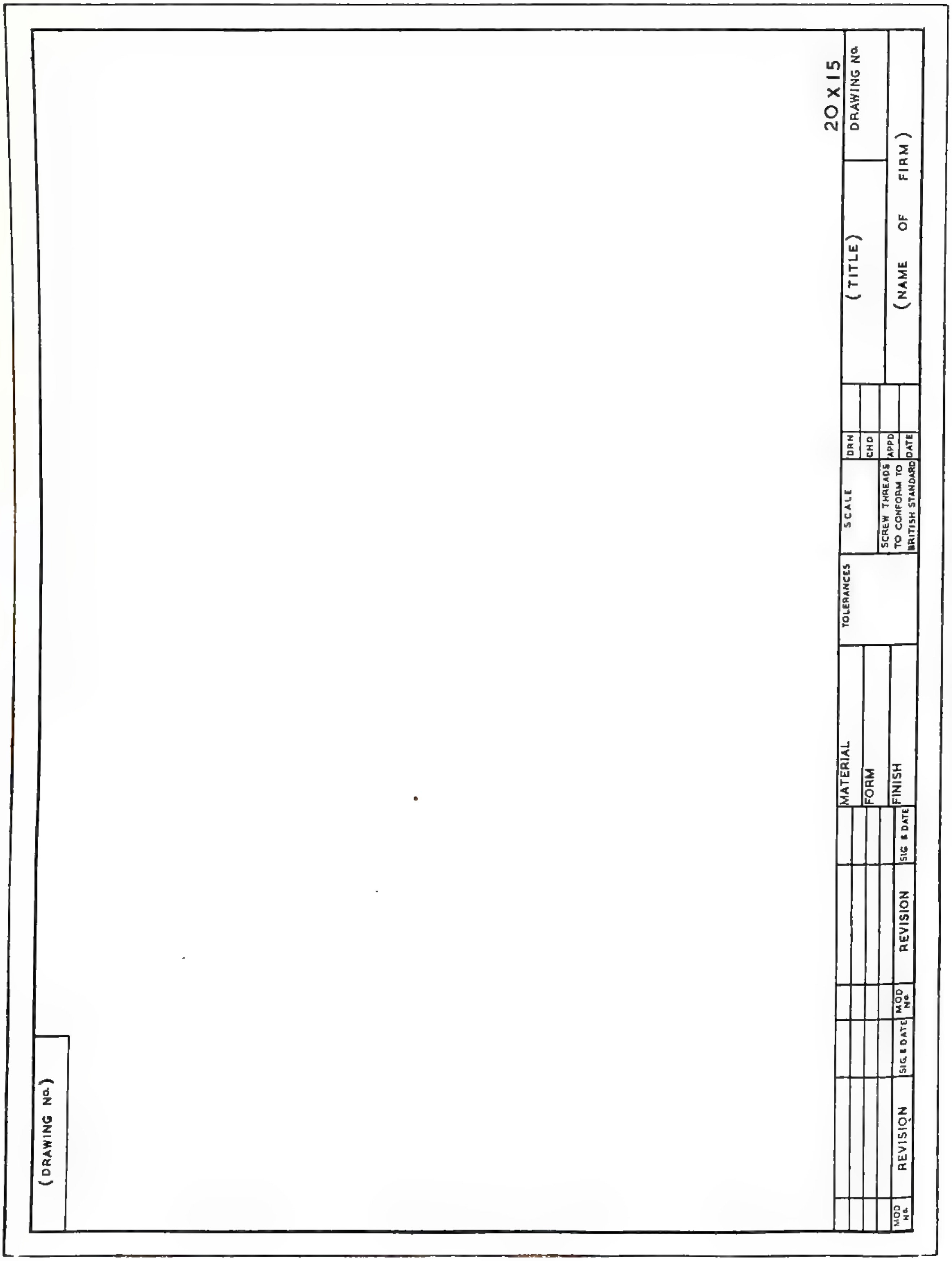


Fig. 2. Typical layout of a drawing sheet

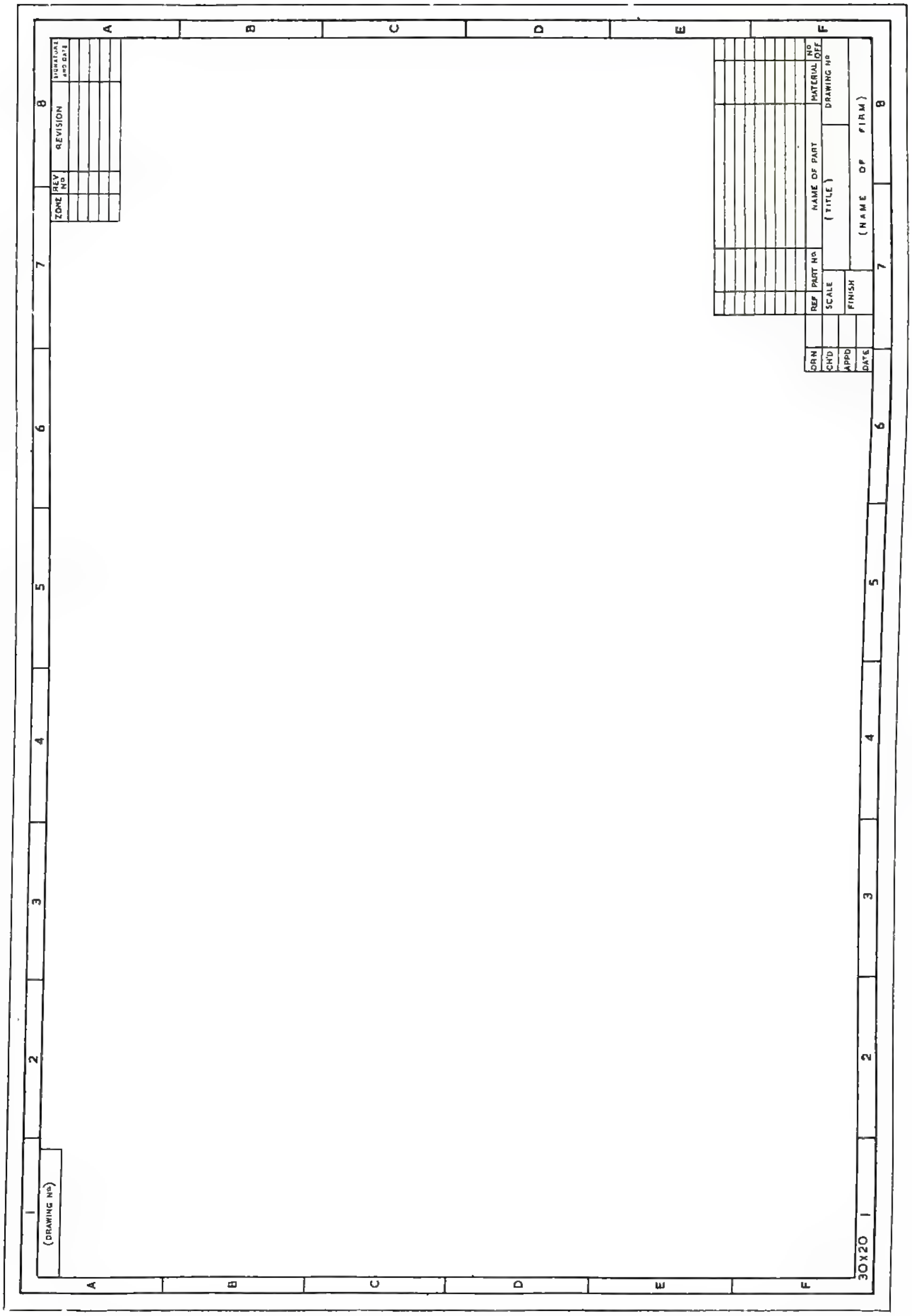


Fig. 3. Typical layout of a drawing sheet

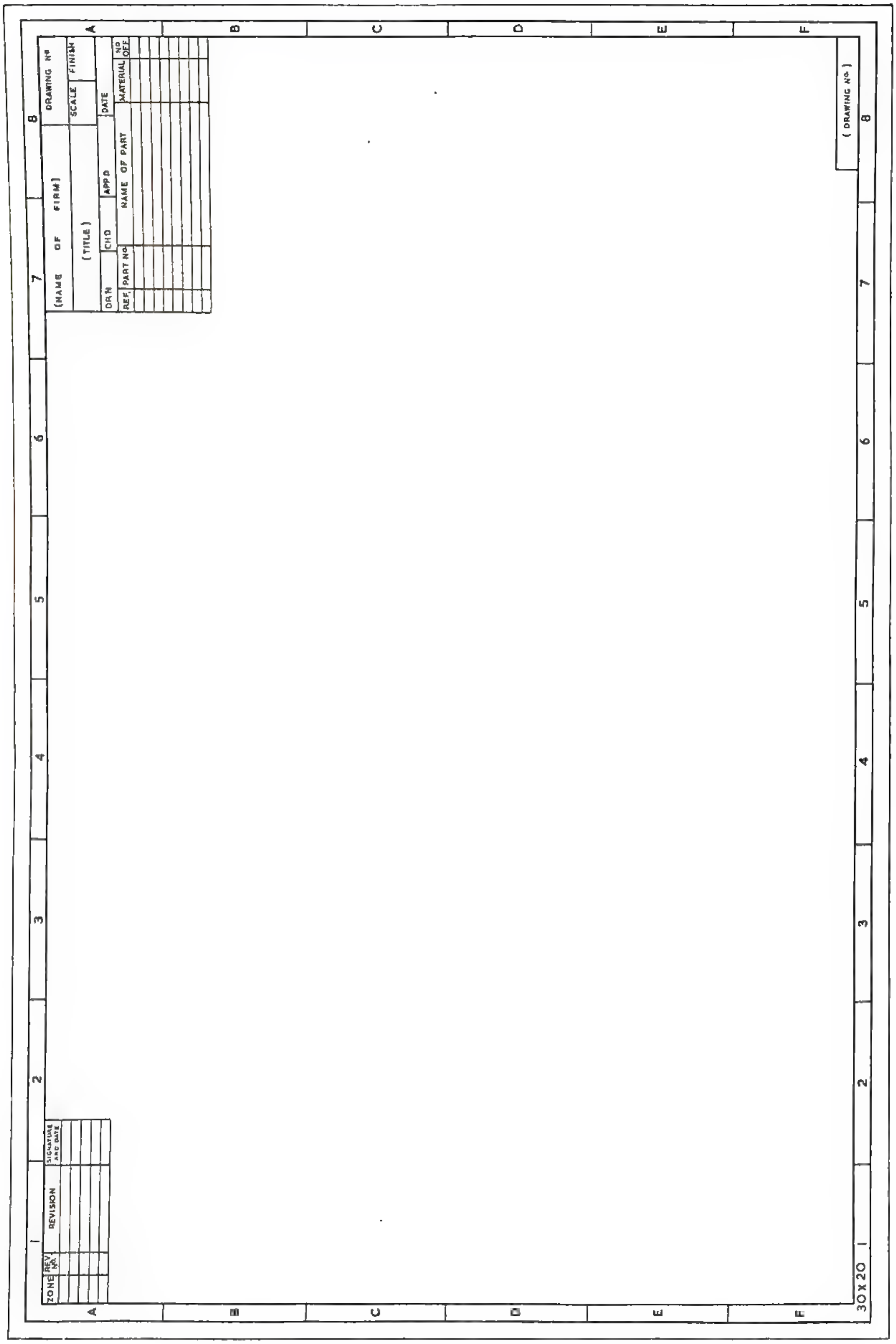


Fig. 4. Typical layout of a drawing sheet

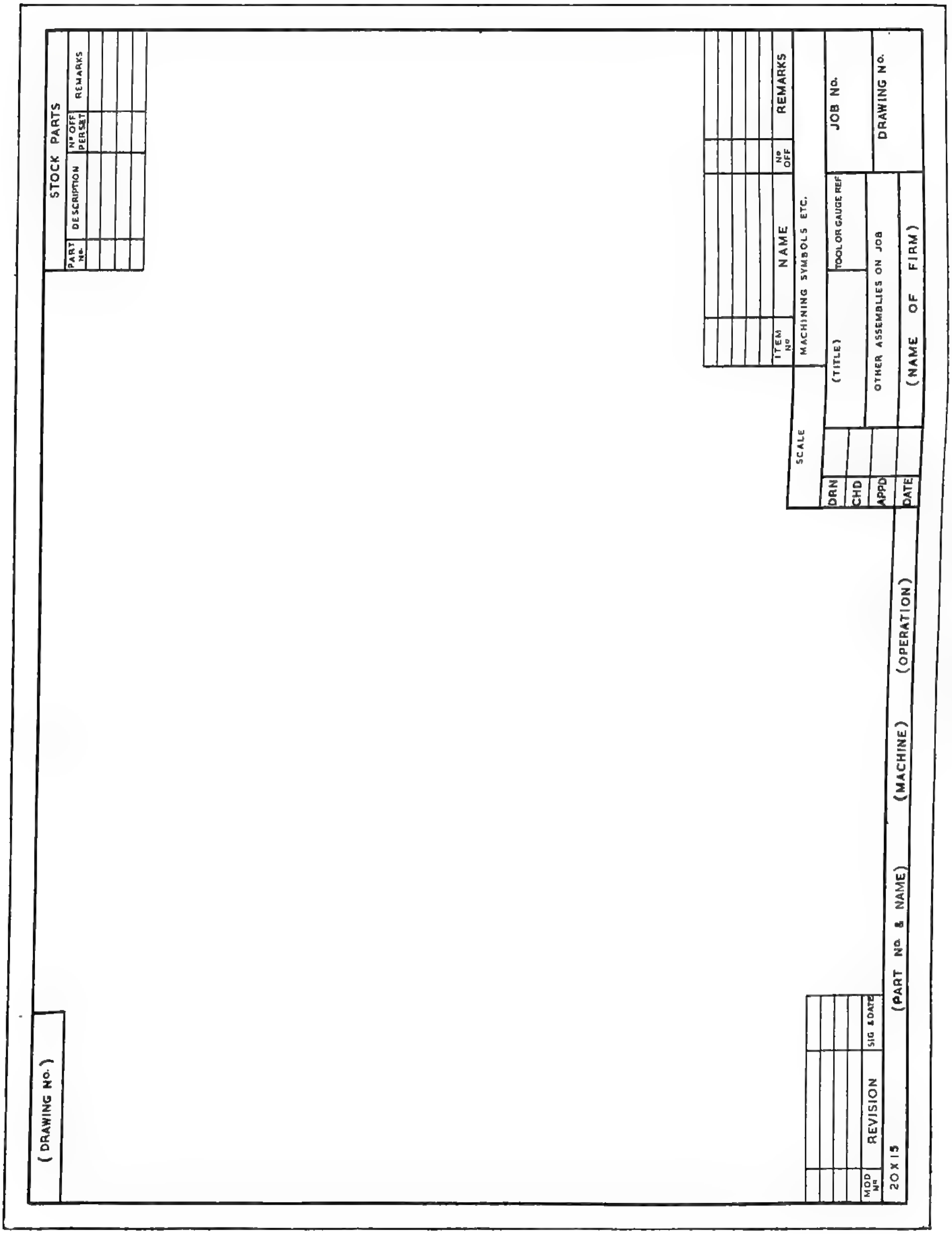


Fig. 5. Typical layout of a drawing sheet

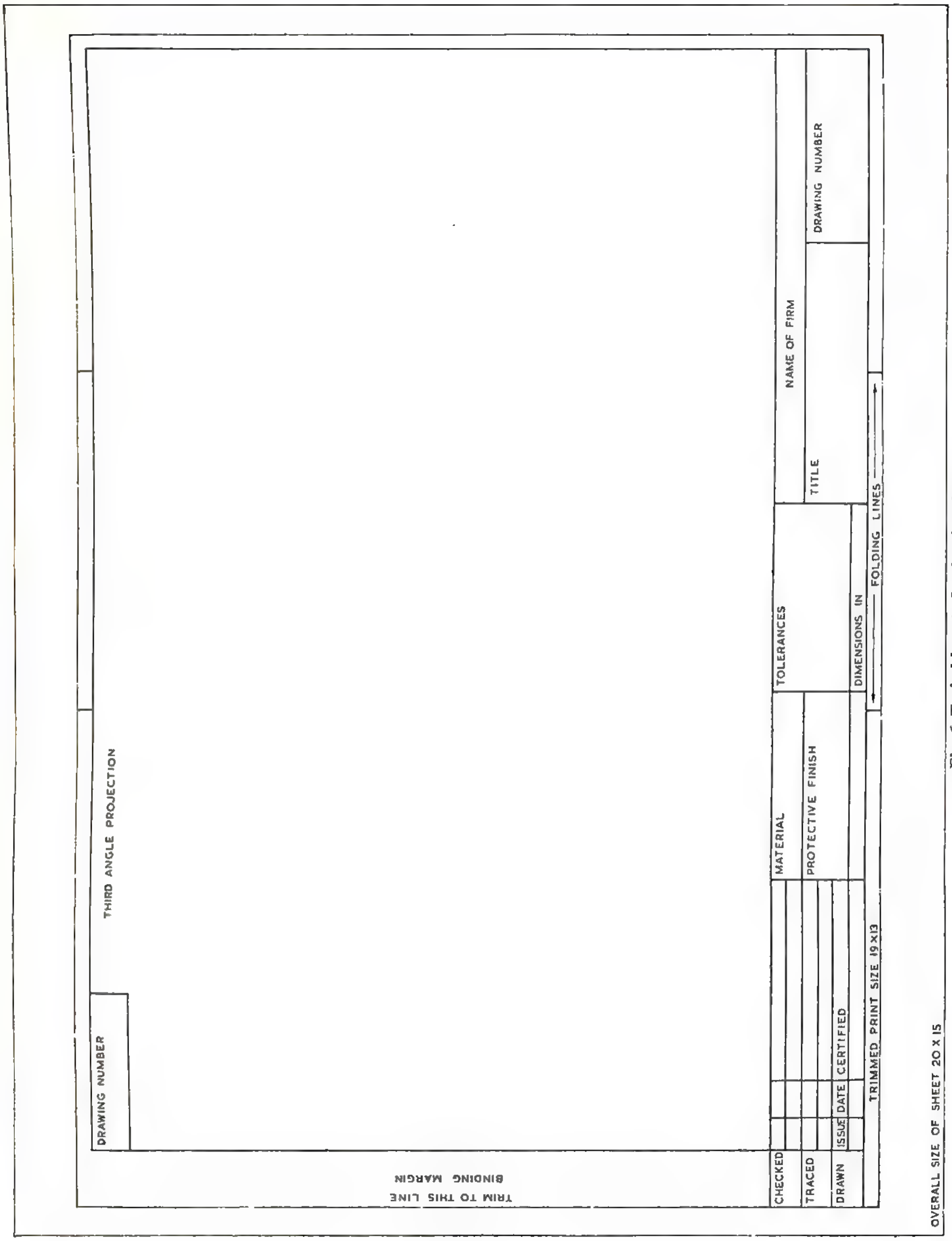


Fig. 6. Typical layout of a drawing sheet

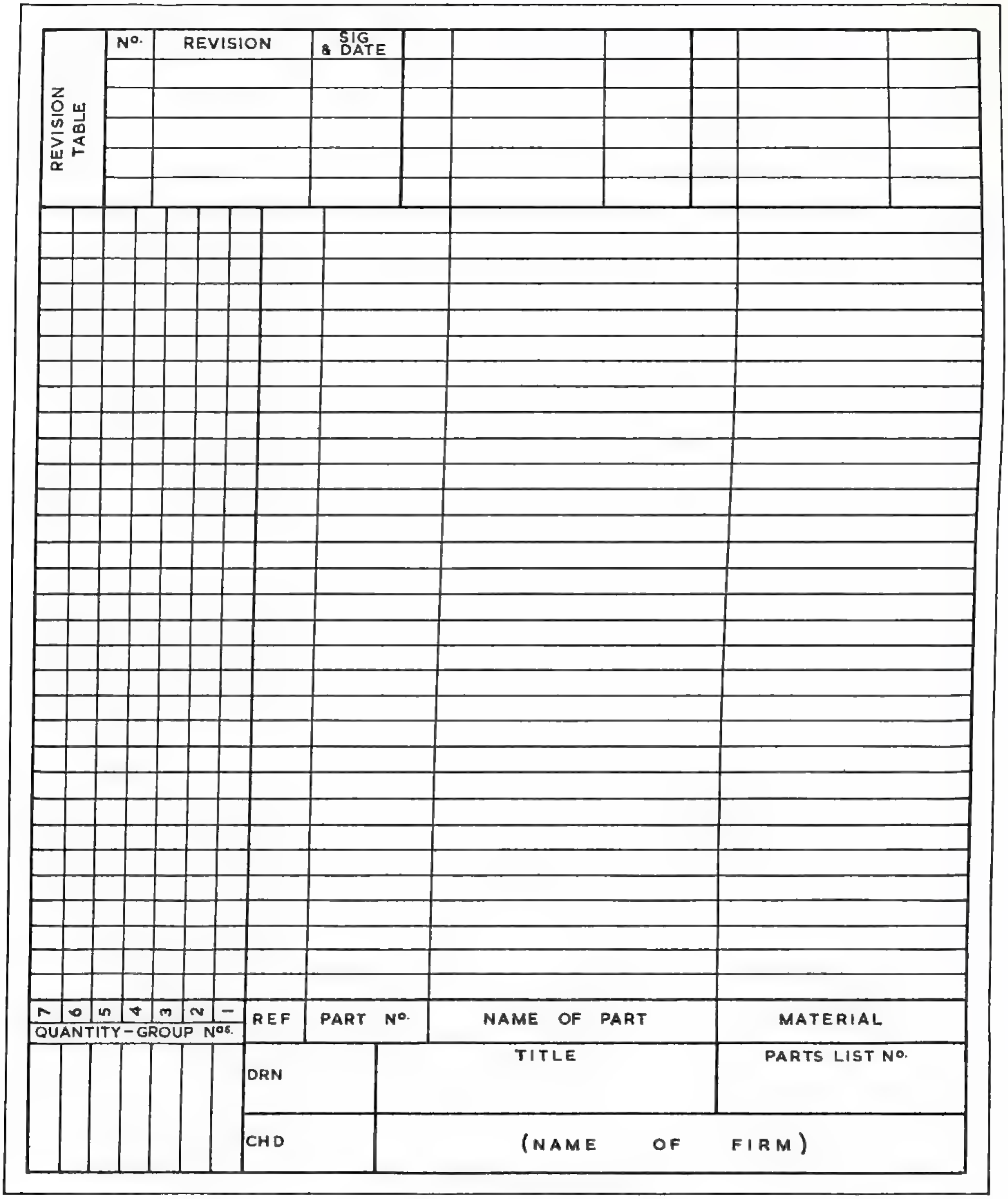


Fig. 7. Typical layout of a parts list

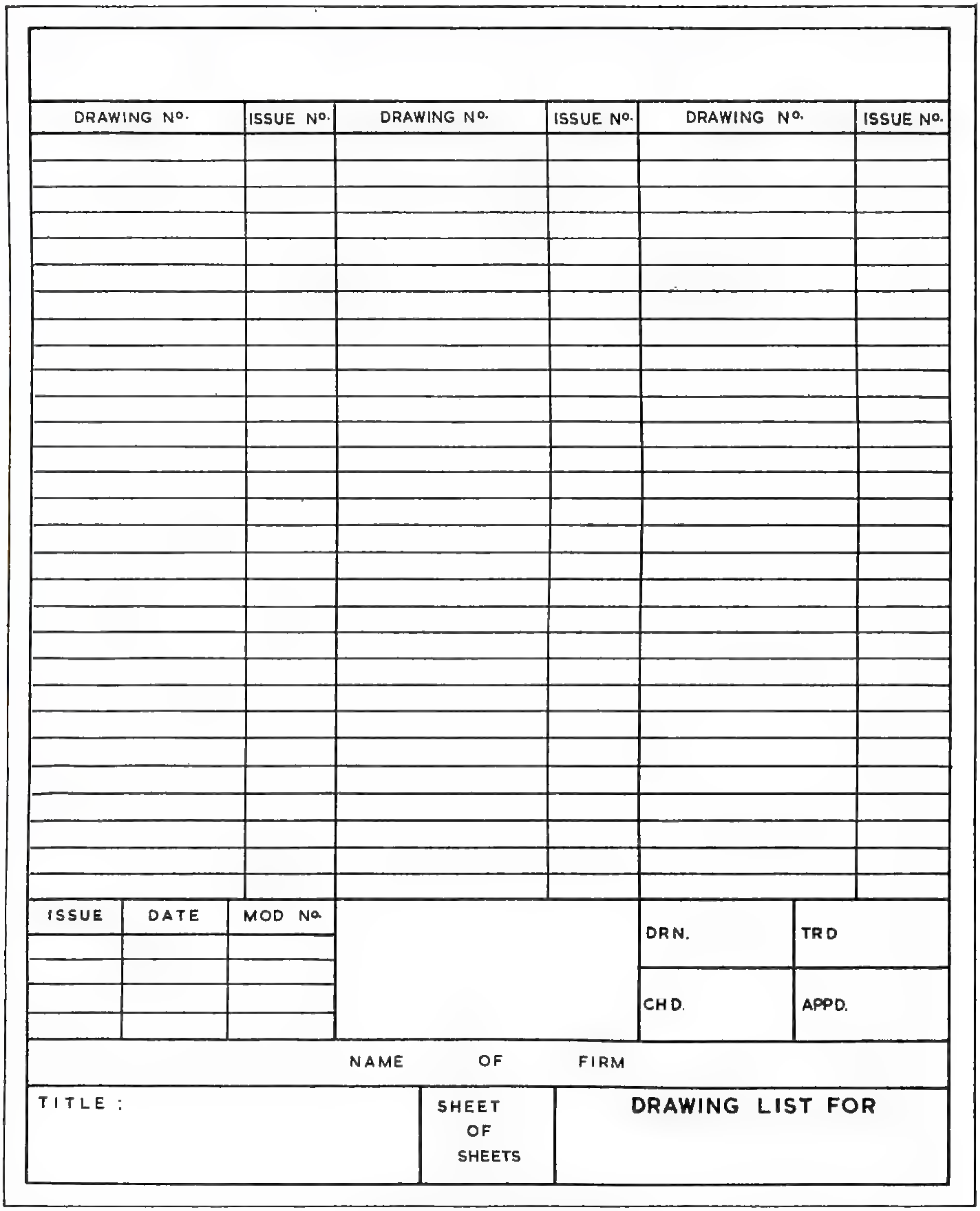


Fig. 8. Typical layout of a drawing list

CLAUSE 2 (*cont.*)

g. Grid system or zoning. In order that a particular dimension or feature on a large drawing can be readily located, it is sometimes desirable to use a grid reference system based on numbered and lettered divisions in the margins (see Figs. 3 and 4).

h. Revisions. It is most important that all revisions should be indicated on the drawing, and that each new issue be identified by a change in the date, issue number, or letter. One convenient method of recording a revision is a table, on the face of the drawing, in which details of the revisions are given. Suitable tables for recording such revisions are shown in Figs. 2, 3, 4 and 5.

The marginal grid reference may be used to identify the position of the revision and the appropriate zone reference quoted in the revision table, e.g. C4, etc. A further method of identification is the use of a revision letter or number enclosed within a circle or square which is placed in proximity to the revised dimension or detail. Reference is made in the revision table or elsewhere to this number or letter. In either case complete details of the change made should be recorded either on the drawing itself or in some other appropriate form.

Where interchangeability of a part is not affected, either in the dimensional or any other sense, the drawing number or part number should not be changed. If interchangeability is affected, a new drawing number or a new part number should be given to the detail concerned.

In conjunction with revisions to drawings it is usual to issue a form or note to record the change, to ensure the replacement or correction of existing prints, and to give instructions concerning any existing parts or work in progress.

j. Assembly drawings. Where a number of drawings are required to detail a complete design, an assembly drawing is necessary. Such drawings will show the project to a convenient scale, and the drawing or part numbers, which deal with each part or section in detail, may be recorded in suitable tabular form on the drawing or on a separate sheet.

k. Key plan. An alternative method, particularly applicable to structural work, is to include on each individual sheet of a series of drawings a small key plan or elevation, or both, conveniently placed preferably near to the title block, indicating in bold lines to which part of the whole work the particular sheet refers. An example is given in Fig. 9.

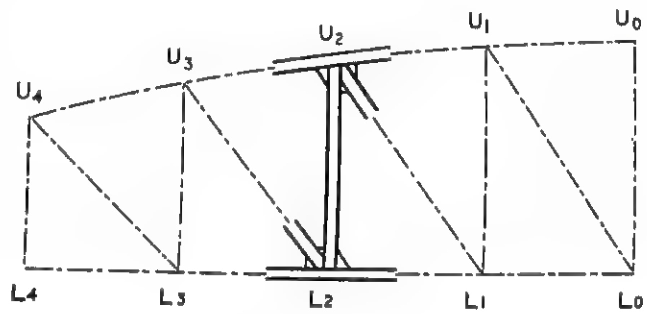


Fig. 9. Key plan

m. Grouping of parts. When several parts are shown on one drawing they should preferably be grouped according to the process involved (e.g. parts machined from bar material, castings, welded structures, sheet metal work, etc.).

n. Orientation of views. A drawing should show an article in the clearest possible manner. In general, the main view of a part should have the same orientation as that shown on the assembly drawing.

NOTE. In most cases assemblies are drawn forward to the left. As a rule, however, ship and marine engineering drawings are made forward to the right, aft to the left.

3. NUMBERING AND REFERENCING

a. Numbering. The system of numbering drawings will be a matter for individual firms or departments but, in general, the following rules should be observed :

(i) A register, book or master file should be used for the systematic allocation of drawing numbers.

(ii) When a given part is changed so as to be no longer interchangeable with the previous design, either in the dimensional sense or for any other reason, a new drawing or part number should be given, as indicated in Clause 2 *h* (above).

(iii) Any given part, although it may be used in various models, types or sizes of mechanism, should always retain the original drawing number or part number and should be detailed on one drawing only.

(iv) Wherever possible, right and left hand parts or assemblies should have consecutive drawing numbers or part numbers.

b. Referencing. Where assembly drawings comprise a large number of parts it is recommended that part reference numbers should be arranged as far as possible in vertical columns or horizontal rows in the most convenient groupings outside the drawn views, with leader lines indicating the respective parts by means of arrow heads or dots.

These part reference numbers should be ringed with a circle or other geometrical figure, to facilitate the location of the numbers when reading the drawing.

4. SCALES

a. Scale. All drawings should be drawn to scale and the scale used stated on the drawing. Drawings of details drawn larger than full size should, where practicable, include an undimensioned view to the actual size. This view may be three dimensional.

Where it is necessary or desirable to indicate that a particular dimension is not to scale, the abbreviation NTS should be added, or the dimension underlined—preferably with a wavy line (see Fig. 11).

When a drawing is to be reproduced in reduced or enlarged form, it is desirable to add a drawn scale.

b. Recommended scales.

Full size (1/1)	¾ in. = 1 ft (1/16)
Half size (1/2)	½ in. = 1 ft (1/24)
	⅜ in. = 1 ft (1/32)
3 in. = 1 ft (1/4)	¼ in. = 1 ft (1/48)
1½ in. = 1 ft (1/8)	⅛ in. = 1 ft (1/96)
1 in. = 1 ft (1/12)	¹⁄₁₆ in. = 1 ft (1/192)

Where scales greater than full size are required, scale multipliers of 2, 4, 8 and 10 are recommended.

For indicating the scale in the appropriate division of the title block a simple abbreviation is :

Scale 1/1 (for full size).
Scale 1/4 (for quarter size).
Scale 2/1 (for twice full size) and so on.

c. Decimal scales. Where decimal scales are required, scale multipliers and divisors of 2, 5 and 10 are recommended.

d. Dual scales. In the case of framed structures, a convenient and much used practice consists in drawing the layout of the centre lines of members to one scale and superimposing drawings of the details to a larger scale on the intersection points of these centre lines. Other views of the parts detailed are drawn in convenient positions alongside. A case of this kind is illustrated in Fig. 10. The two scales used should be clearly indicated.

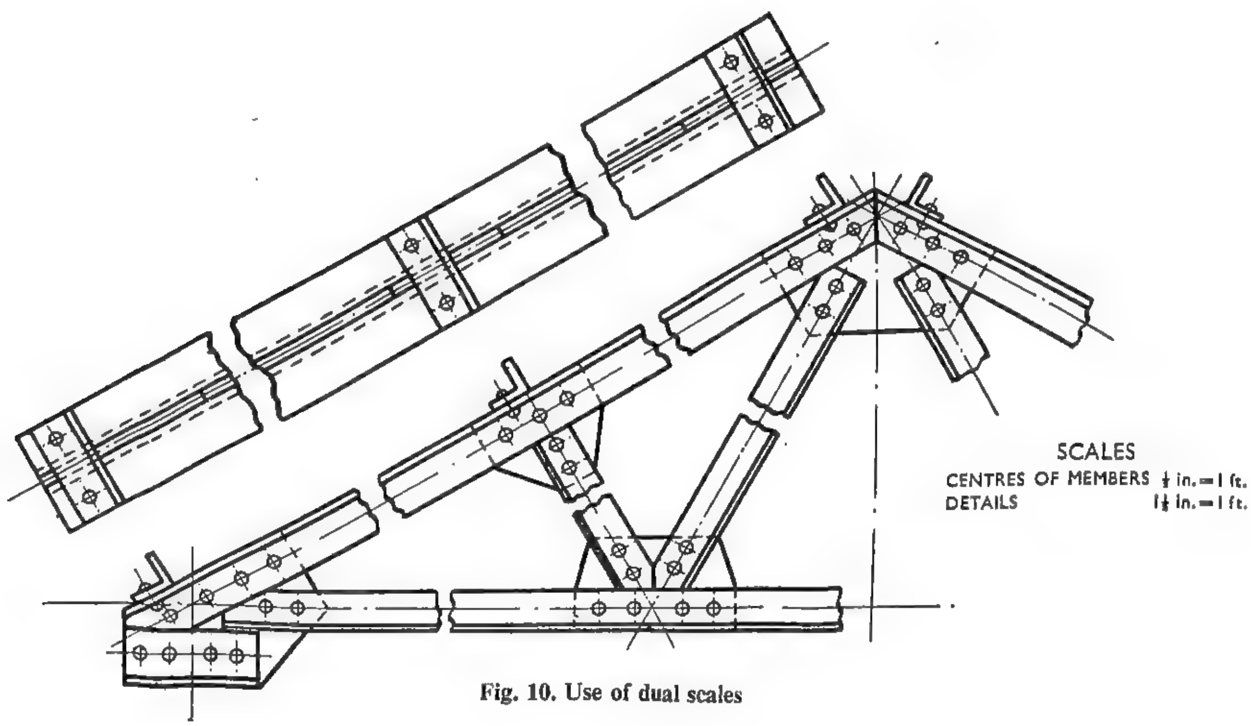
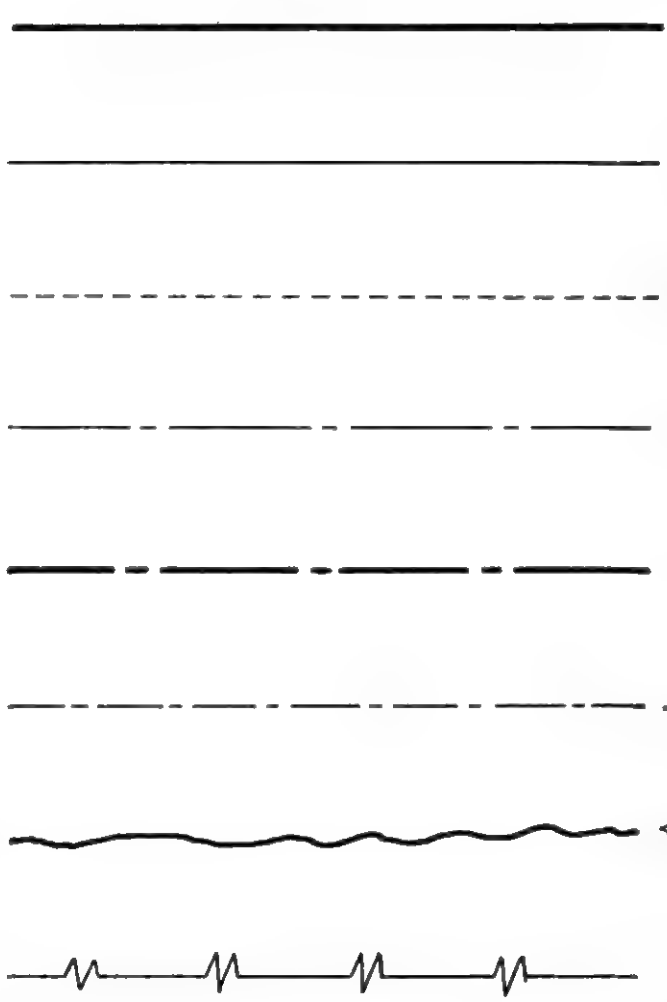


Fig. 10. Use of dual scales

5. TYPES OF LINES

For general engineering drawings, the following types of lines should be used :

Type of line		Example	Application
Continuous (thick)	A		Visible outlines.
Continuous (thin)	B		Dimension lines. Projection or extension lines. Hatching or sectioning. Leader lines for notes.
Short dashes (thin)	C		Hidden details. Portions to be removed.
Long chain (thin)	D		Centre lines. Path lines for indicating movement.
Long chain (thick)	E		Cutting or viewing planes.
Short chain (thin)	F		Developed or false views. Adjacent parts.
Continuous wavy (thick)	G		Irregular boundary lines. Short break lines.
Ruled line and short zig-zags	H		Long break lines.



Lines should be sharp and dense to obtain good reproduction.

Lines specified as thick should be from two to three times the thickness of lines specified as thin.

Centre lines should project for a short distance beyond the outline to which they refer, but where necessary to aid dimensioning or to correlate views, they may be extended.

Typical applications of recommended types of lines are shown in Fig. 11.

CLAUSE 5 (*cont.*)

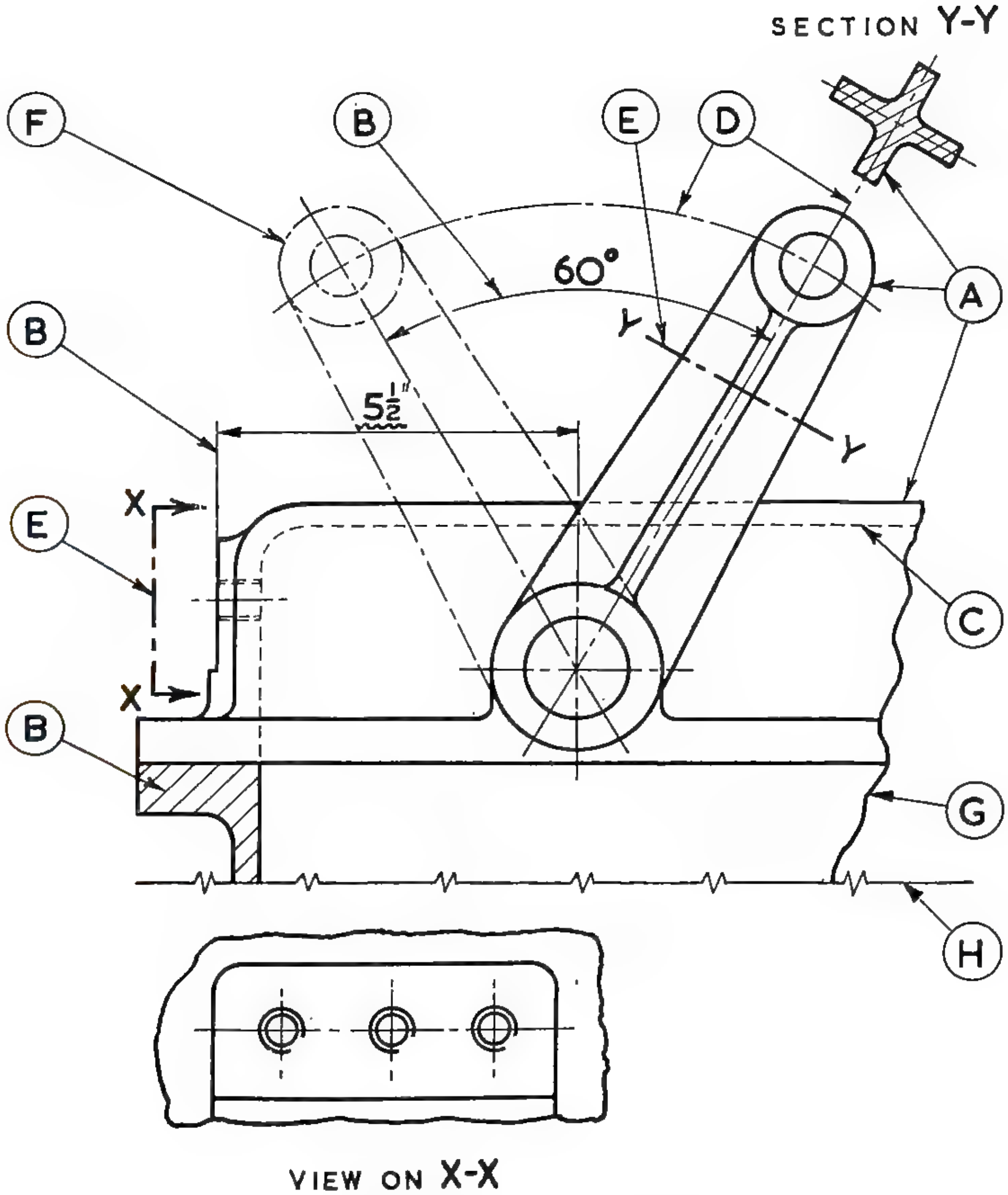


Fig. 11. Typical applications of recommended types of lines

6. PROJECTION

a. Systems of projection. Two systems of projection, known respectively as FIRST ANGLE and THIRD ANGLE are acceptable as British Standards.

In FIRST ANGLE PROJECTION each view shows what would be seen by looking on the far side of an adjacent view (see Fig. 12).

In THIRD ANGLE PROJECTION each view shows what would be seen by looking on the near side of an adjacent view (see Fig. 13).

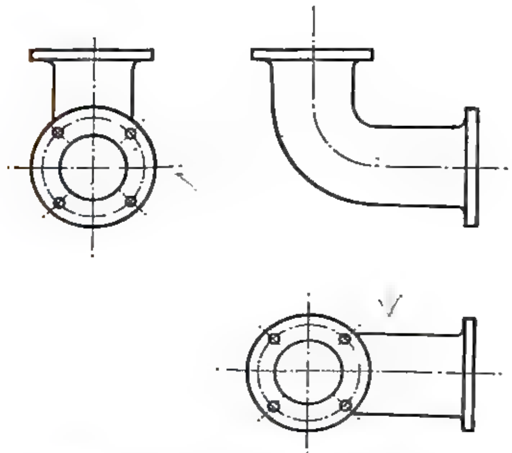


Fig. 12. Pipe bend, first angle projection

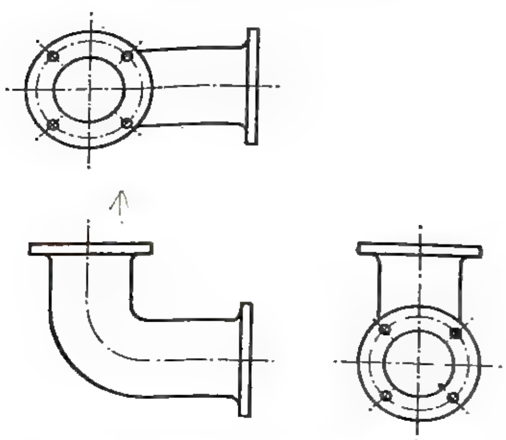


Fig. 13. Pipe bend, third angle projection

b. Marking of drawings. The drawings should be marked to indicate the method of projection used; otherwise, the directions in which the views are taken should be clearly indicated.

c. Long objects. With first angle projection for long objects, it is not uncommon to place an end view adjacent to the face which it represents. The practice should be indicated by a note when it is adopted; a suitable note would be ' view in direction of arrow A '.

d. Number of views. The number of views should be the minimum necessary to ensure that there will be no misunderstanding. Views should be selected to give as few hidden lines as possible. Scrap or local views may be used to define particular features where a complete view of the object is not necessary (see Fig. 11).

e. Auxiliary views. Objects having inclined faces may have such faces projected to show the true shape of the inclined surface (see Fig. 14).

f. Space between views. Ample space should be provided between views to avoid crowding of dimensions or notes.

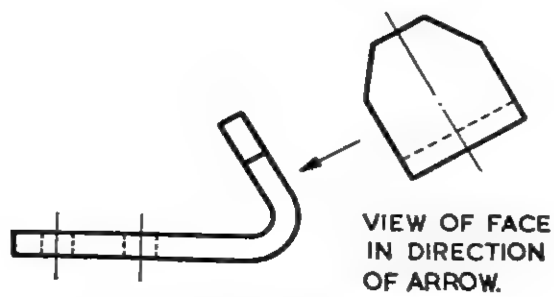


Fig. 14. Projection of inclined faces

7. LETTERING

The objective should be to employ uniform letters and figures which can be produced with reasonable rapidity and which will ensure good and legible reproductions from either pencil or ink originals.

On large drawings which are to be reduced for illustrative or other purposes, the lettering should be large enough to be legible when reduced.

Sloping or vertical letters and figures are suitable for general use. However, vertical characters are recommended for drawing numbers, titles and reference numbers. The specimens reproduced in Fig. 15 are given as a guide.

Fig. 15. Types of lettering

8. SECTIONING AND SECTIONAL VIEWS

a. *Sectional views.* A sectional view represents that part of an object which remains after a portion is assumed to have been cut or broken away and removed. The exposed cut or broken surfaces should be indicated by section lining (hatching), except for relatively simple cases where the construction is obvious.

Section lining should be made with thin parallel lines (type B, Clause 5, page 17), usually drawn at an angle of 45° to the edges of the drawing sheet and suitably spaced in relation to the area to be covered. Where large areas of sectioning have to be shown, it is recommended that the edges only be sectioned as shown in Fig. 16.

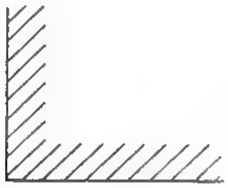


Fig. 16. Section lining of large areas

If the shape or position of the section would bring 45° sectioning parallel or nearly parallel to one of the sides, another angle may be chosen (see Fig. 17).

In all views showing sections of the same part, the sectioning should be similar in direction and spacing.

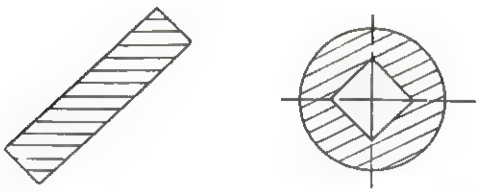


Fig. 17. Special section lining

Adjacent parts should be sectioned in different directions or to a different pitch (see Fig. 18).

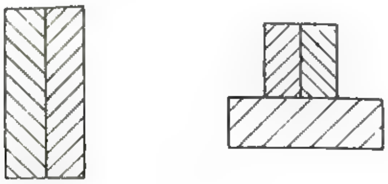


Fig. 18. Section lining for adjacent parts

Where it is necessary to insert dimensions or lettering on a sectioned area, section lines should be interrupted (see Fig. 19).

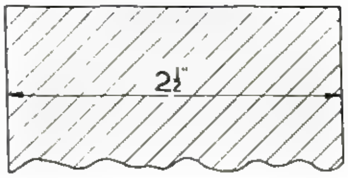


Fig. 19. Section lining interrupted for dimensions

b. *Location of cutting planes.* The position of a cutting plane should, where necessary, be indicated on one or more views of an object by thick long chain lines (Clause 5, type E) with letters in bold type, and, where required, arrows at the ends pointing in the direction in which the object is viewed in section (see Figs. 20, 21 and 22).

The following examples illustrate the methods which should be adopted :

Fig. 20. A section in one plane not on a centre line.

Fig. 21. A section in one plane along a centre line.

Fig. 22. The path of a section in more than one plane. The path should be indicated by thick long chain lines (see type E, Clause 5, page 17).

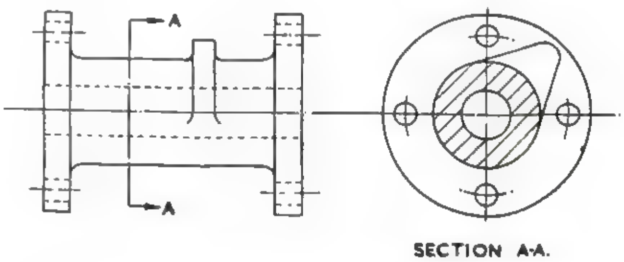


Fig. 20. Section in one plane not on centre line

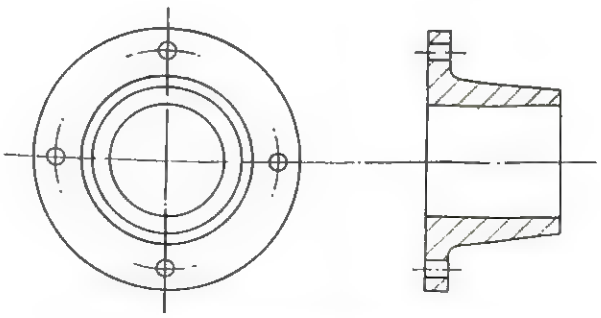


Fig. 21. Section in one plane along centre line

CLAUSE 8 (*cont.*)

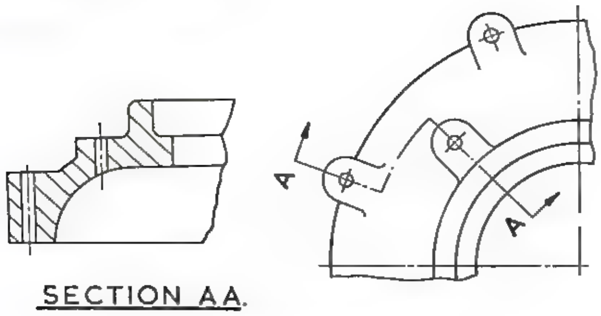


Fig. 22. Section in two planes

c. Half sections. Objects which are symmetrical about a centre line may be drawn having one half in outside view and the other half in section (see Fig. 23).

Hidden lines behind the cutting plane (see right hand half section, Fig. 23) should be omitted unless required for the definition of the object.

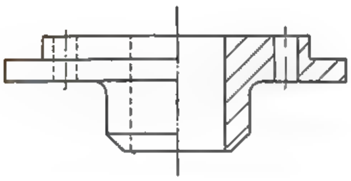


Fig. 23. Detail in half section

Hidden lines on the unsectioned side (see left hand side, Fig. 23) may be shown if needed for dimensioning or clarity. In assemblies it is customary not to show hidden lines on the unsectioned side (see Fig. 24).

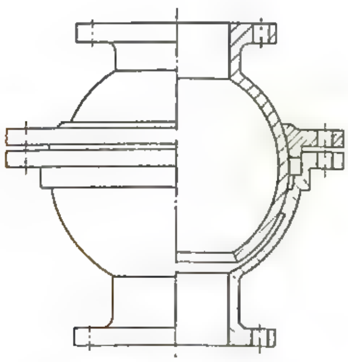


Fig. 24. Assembly in half section

d. Part sections. Part sections may be taken at convenient places to show detail which would otherwise be hidden, the boundary of such sections being shown by an irregular boundary line (type G, Clause 5, page 17) as shown in Fig. 25.

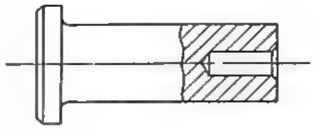


Fig. 25. Part section

e. Revolved and removed sections. Revolved sections show the shape of the cross section on the actual view of the part, the cutting plane being revolved in position (see Fig. 26).

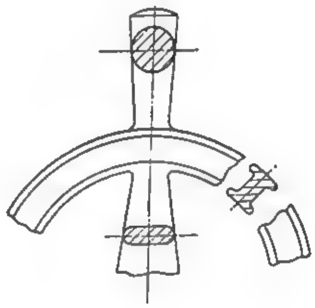


Fig. 26. Revolved sections

Removed sections are similar to revolved sections except that the cross section is removed from the actual view of the part (see Fig. 27).

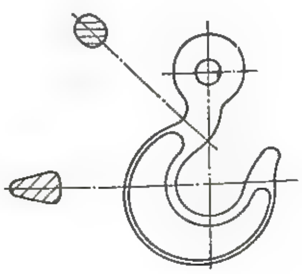


Fig. 27. Removed sections

CLAUSE 8 (*cont.*)

f. Thin sections. Sections which are too thin for line sectioning may be shown solid, e.g. structural shapes, sheet metal, packing, gaskets, etc. Where two or more adjacent parts are shown solid, a space should be left between them (see Fig. 28).

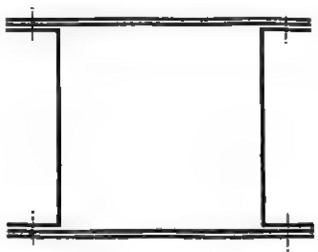


Fig. 28. Thin sections

g. Exceptions. Exceptions to the general rule for indicating sections should be made where the cutting plane passes longitudinally through ribs, shafts, bolts, nuts, rods, rivets, keys, pins and similar parts. These should be shown by outside views and not in section (see Fig. 29).

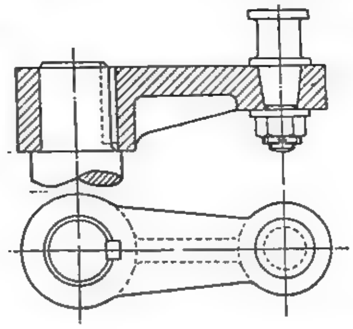


Fig. 29. Rib, shaft, and crank pin not shown in section

9. CONVENTIONAL REPRESENTATION OF COMMON FEATURES AND MATERIALS

a. Common features. Conventional representation is adopted in cases where complete delineation of the part would involve unnecessary drawing-time or space. Typical examples are shown in Figs. 30 to 35.

Where the conventional representation given is not considered adequate, a more detailed view may be shown.

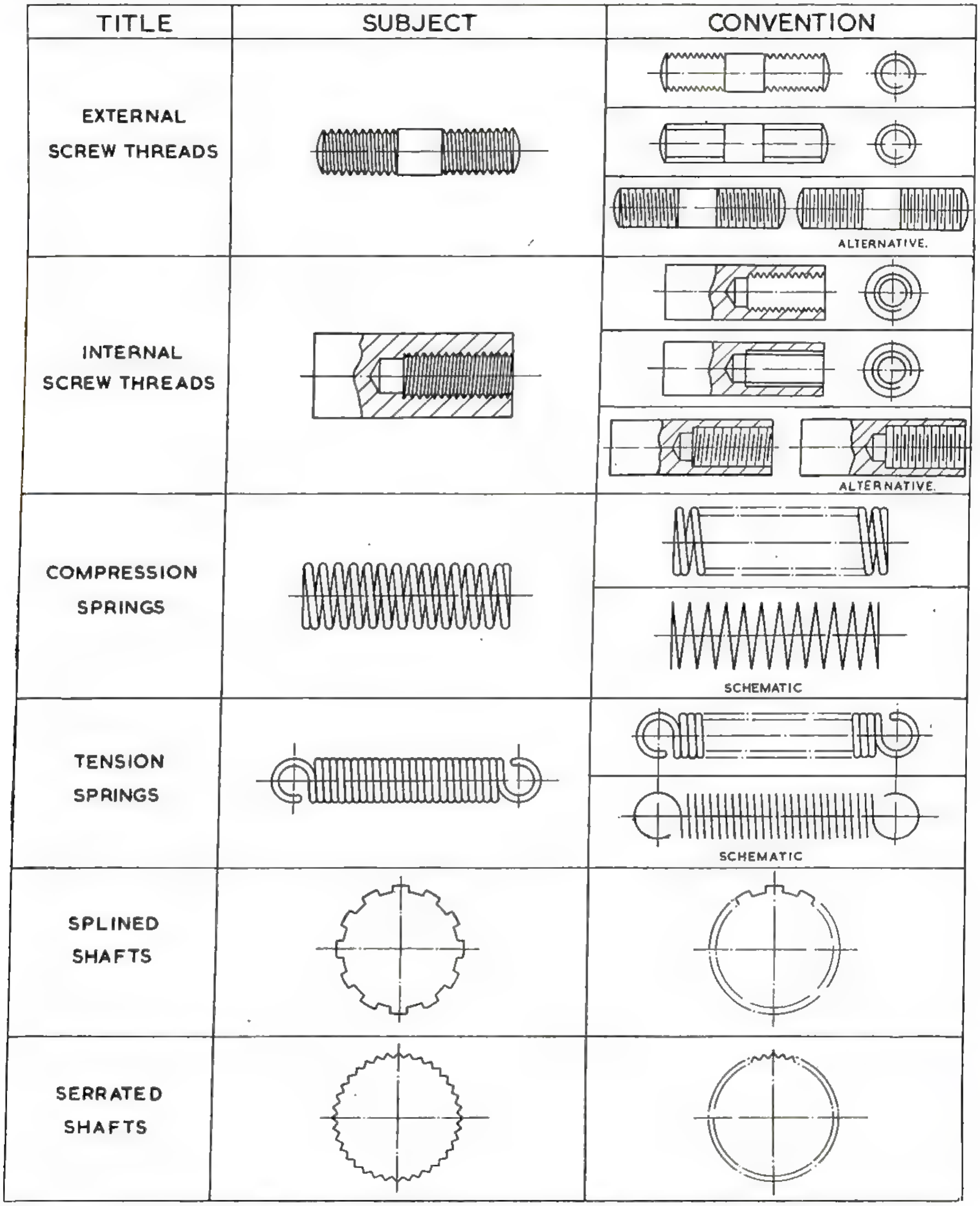


Fig. 30. Conventional representation of common features

CLAUSE 9 (*cont.*)

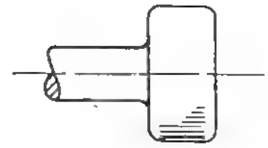
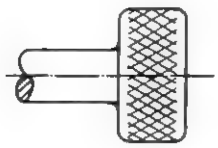
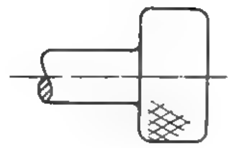
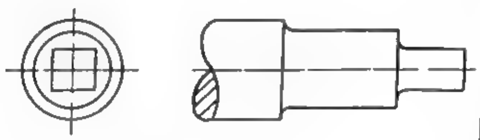
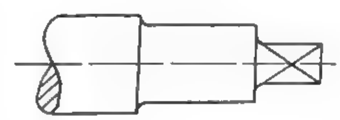
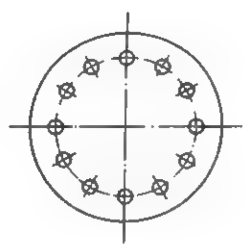
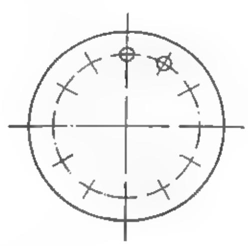
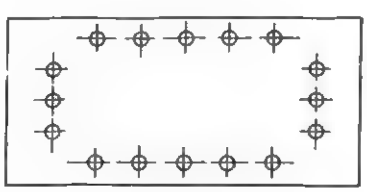
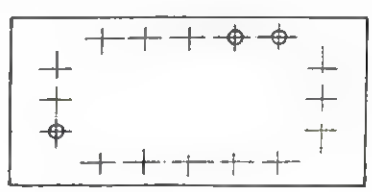
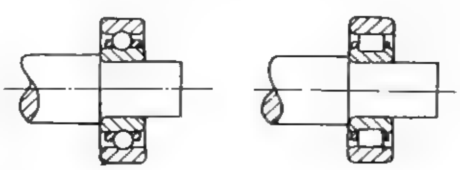
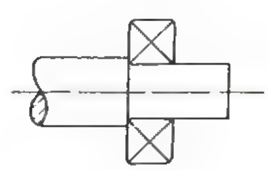
TITLE	SUBJECT	CONVENTION
STRAIGHT KNURLING.		
DIAMOND KNURLING.		
SQUARE ON SHAFT.		
HOLES ON CIRCULAR PITCH.		
HOLES ON LINEAR PITCH.		
BEARINGS.		

Fig. 31. Conventional representation of common features

CLAUSE 9 (*cont.*)

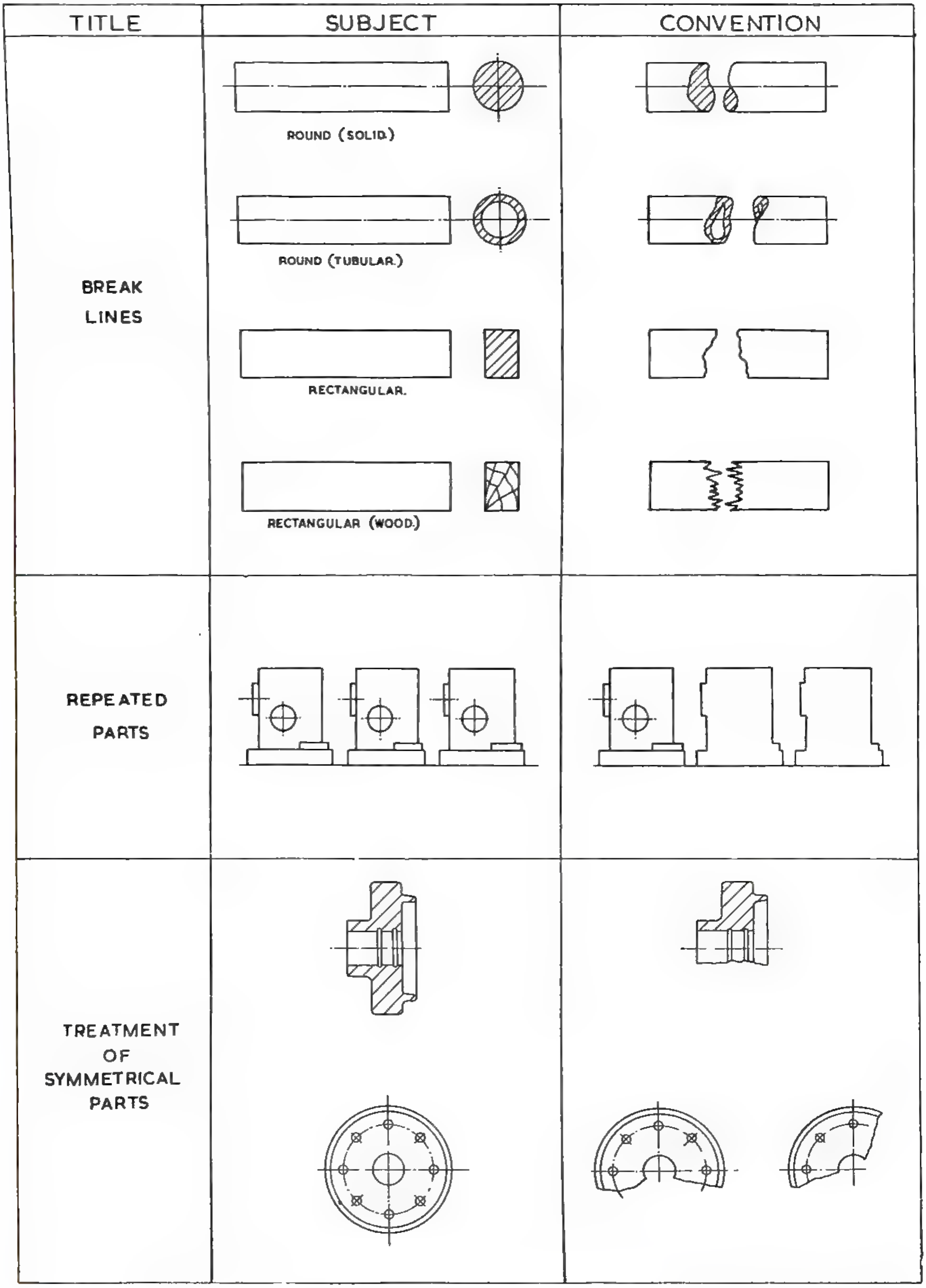


Fig. 32. Conventional representation of common features

CLAUSE 9 (*cont.*)

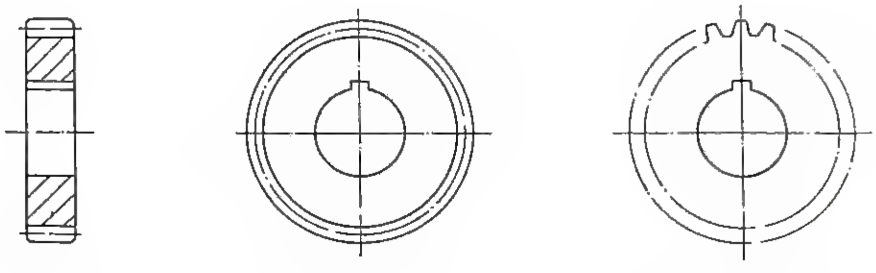
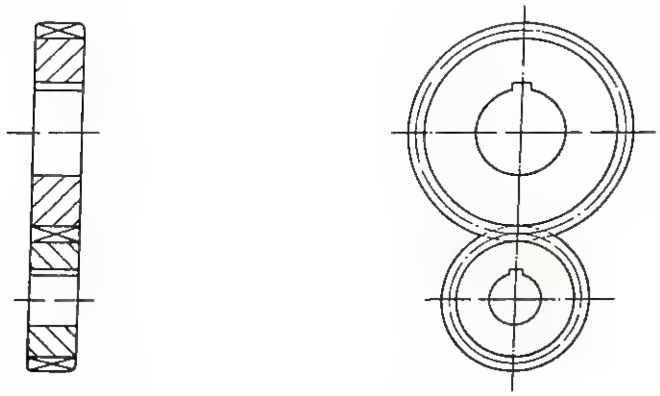
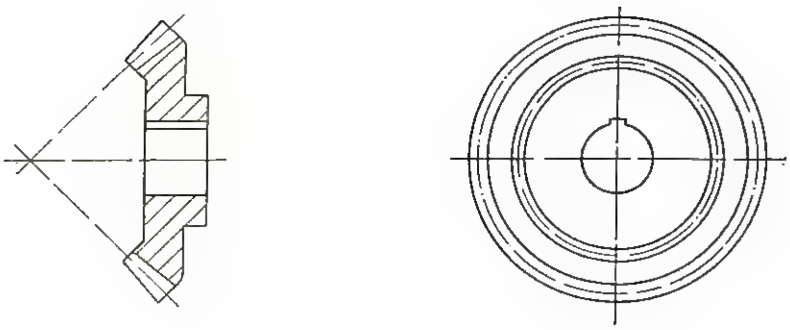
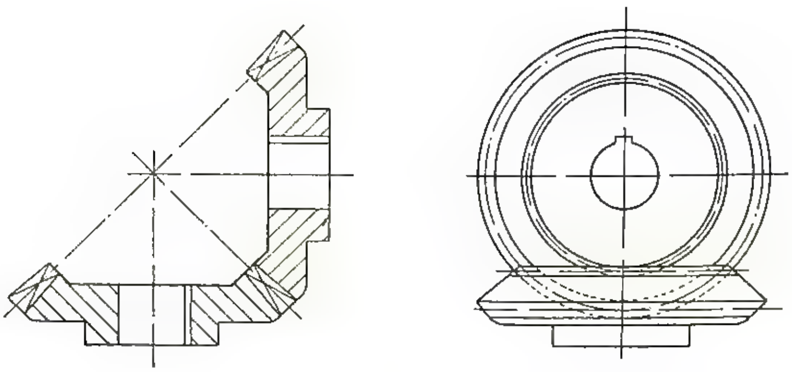
TITLE	CONVENTION
SPUR GEAR (DETAIL.)	 ALTERNATIVE.
SPUR GEARS (ASSEMBLY.)	
BEVEL GEAR (DETAIL.)	
BEVEL GEARS (ASSEMBLY.)	

Fig. 33. Conventional representation of common features

CLAUSE 9 (*cont.*)

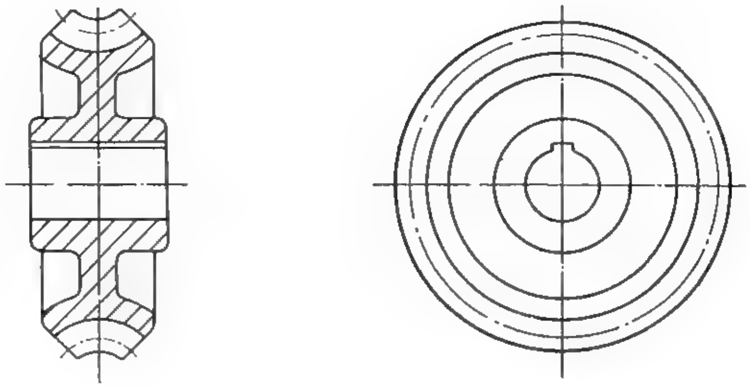
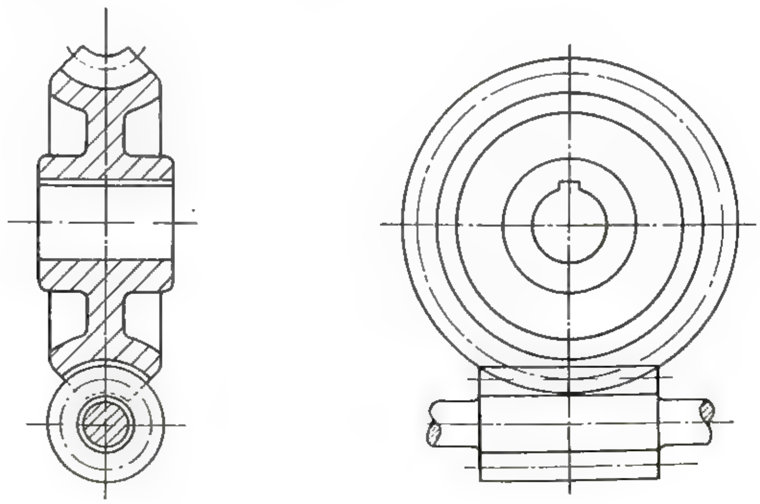
TITLE	CONVENTION
WORMWHEEL (DETAIL)	
WORM (DETAIL)	
WORM AND WORMWHEEL (ASSEMBLY)	

Fig. 34. Conventional representation of common features

CLAUSE 9 (*cont.*)

TITLE	CONVENTION
ELECTRIC WINDING	
FINE WIRE MESH	
COARSE WIRE MESH	
PERFORATED SHEET	

Fig. 35. Conventional representation of common features

b. *Materials.* (i) *Sectioning.* In view of the variety of materials used it is undesirable to rely on the various conventions of section lining to differentiate between different materials.

It is therefore recommended that, with the exception of insulation, glass, wood, concrete and water, ordinary section lining be used in all cases where materials are shown in section.

Fig. 37 illustrates recommended methods of showing insulation, glass, wood, concrete and water.

(ii) *Colouring.* Where it is desired to indicate materials in section by colours or coloured lines, the following are recommended :

Material	*Colour*
Cast iron	Payne's grey
Wrought iron	Prussian blue
Steel	Purple
Brass, phosphor bronze and gunmetal	Light yellow
Copper	Crimson lake
Aluminium, tin, white metals and light alloys	Light green
Brickwork	Vermilion
Concrete	Light green
Earth, rock	Sepia
Timber	Burnt sienna
Glass	Pale blue wash
Insulation (electrical)	Black

c. *Methods of indicating rivets.* Normally, rivets may be indicated sufficiently by their centre lines in the direction of, and at right angles to, the seams.

In long seams such indication need only be shown at each end, the intermediate rivets being indicated only by the centre lines of the rivet rows in the direction of the seam (see Fig. 36).

Fig. 36. Indication of rivets

d. *Rivet symbols.* Where symbols are used to indicate types of rivet, a key to the symbols should be shown.

CLAUSE 9 (*cont.*)

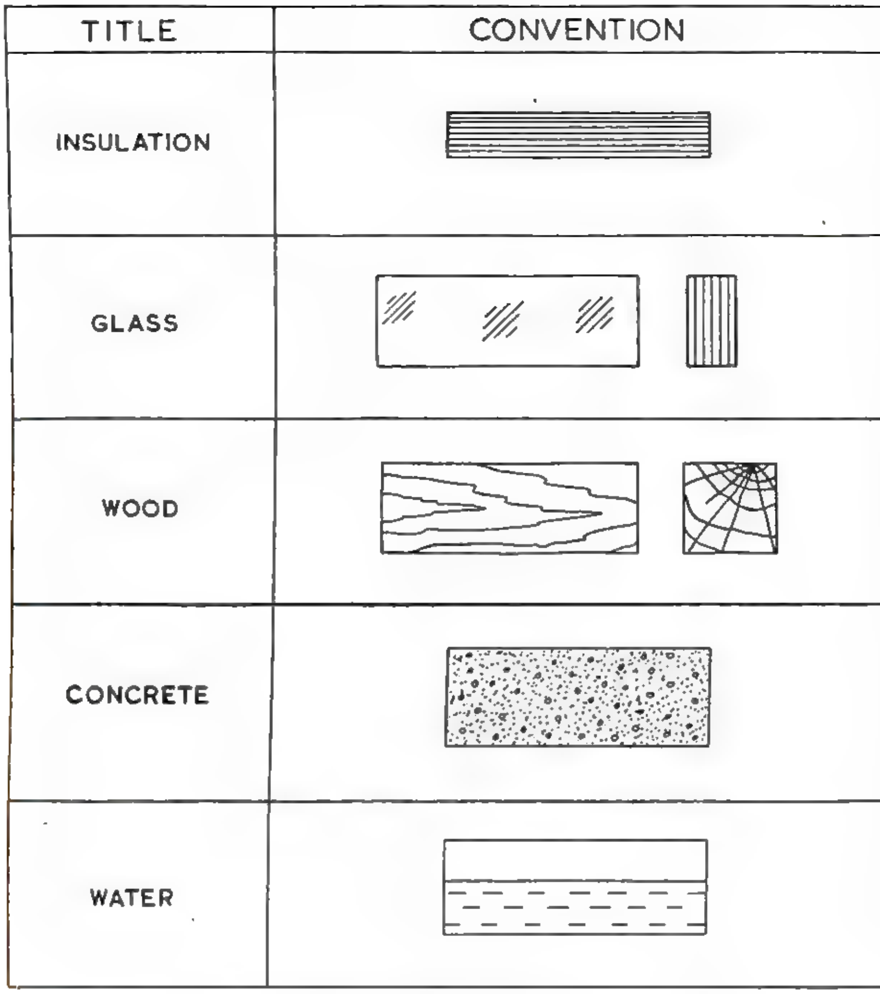


Fig. 37. Conventional representation of materials

e. British Standards for rivets. There are British Standards prescribing the forms and dimensions for many types of rivets, which should be specified where appropriate.

f. Welding symbols. If it is considered desirable to indicate welds on drawings by means of symbols, reference should be made to the appropriate British Standard.*

g. Standard rolled steel sections. With the exception of beams and channels, rolled steel sections are usually designated by the sectional dimensions. Beams and channels may be designated by the nominal depth and the nominal width of flanges and the nominal weight per foot.

Where rolled sections are listed on a drawing they may be designated by symbols or reference letters, as shown in the adjacent table, symbols being preferred.

h. Proprietary sections. Where proprietary sections are specified on a drawing they should be described by the full trade designation as nearly as possible in accordance with the above.

Section	Symbol	Ref. letters	Examples
Beam	I	BSB	18 in. × 6 in. × 55 lb I 18 in. × 6 in. × 55 lb BSB
Channel	[	BSC	17 in. × 4 in. × 44·34 lb [17 in. × 4 in. × 44·34 lb BSC
Angle (equal)	L	BSEA	3 in. × 3 in. × 3/8 in. L 3 in. × 3 in. × 3/8 in. BSEA
Angle (unequal)	L	BSUA	4 in. × 3 in. × 3/8 in. L 4 in. × 3 in. × 3/8 in. BSUA
Tee	T	BST	6 in. × 3 in. × 1/2 in. T 6 in. × 3 in. × 1/2 in. BST
Bulb Angle		BSBA	7 in. × 3½ in. × 15·3 lb 7 in. × 3½ in. × 15·3 lb BSBA
Bulb Plate		BSBP	10 in. × 1/2 in. × 25·66 lb 10 in. × 1/2 in. × 25·66 lb BSBP

* B.S. 499 : Section 7, ' Scheme of symbols for welding '.

10. ABBREVIATIONS FOR USE ON DRAWINGS

The following abbreviations should be used on drawings when required. Abbreviations are the same in the singular and plural. Although capital letters are shown, lower-case letters may be used.

Further recognized abbreviations are listed in other British Standards.

GENERAL ENGINEERING TERMS

Term	*Abbreviation*
Across flats	A/F
Birmingham gauge	BG
Brinell hardness number	BHN
British Standard	BS
Centimetre	CM
Centre line	CL or ℄
Chamfered	CHAM
Cheese head	CH HD
Countersunk	CSK
Counterbore	C'BORE
Cylinder or cylindrical	CYL
Datum system	DATUM
Degree (of angle)	°
Diameter	DIA or ⌀
Diamond pyramid hardness number	DPN
Drawing	DRG
Figure	FIG
Foot	FT (or ′)
Galvanized	GALV
Ground level	GL
Hexagon	HEX
Hydraulic	HYD
Imperial (standard) wire gauge	SWG
Inch	IN. (or ″)
Insulated or insulation	INSUL
Internal diameter	I/D
Kilogram	KG
Left hand	LH
Machine	M/C
Machined	M/CD
Machinery	M/CY
Material	MATL
Maximum	MAX
Metre	M
Millimetre	MM
Minimum	MIN
Minute (of angle)	′

Term	*Abbreviation*	
Not to scale	NTS	
Number	NO.	
Outside diameter	O/D	
Pattern number	PATT NO.	
Per	/	
Pitch circle diameter	PCD	
Pneumatic	PNEU	
Pound	LB	
Radius	RAD or R	
Revolutions per minute	RPM	
Right hand	RH	
Round head	RD HD	
Screw threads:		
British Association	BA	
British Standard fine	BS FINE	
British Standard pipe	BS PIPE	
British Standard Whitworth	BSW	
British Standard cycle	BS CYCLE	
British Standard conduit	BS COND	
Système international	SI	
Unified thread form:	UN	Preceded by nominal diameter and number of threads per inch.
Coarse	UNC	
Fine	UNF	
Special	UNS	
Screwed	SCR	
Second (of angle)	″	
Sheet, when preceding a material	SH	
Sketch	SK	
Sluice or stop valve	SV	
Specification	SPEC	
Spot faced	S'FACE	
Square	SQ	
Square inch	SQ IN or □″	
Standard	STD	
Threads per inch	TPI	
Undercut	U'CUT	
Weight	WT	

TERMS RELATING TO DIMENSIONS AND TOLERANCES

Term	*Abbreviation*
Basic dimension	BASIC
Datum Datum system Datum dimension	DATUM
True position, or true profile, dimension in conjunction with positional, or profile tolerances	TP
Angularity tolerance	ANG TOL
Concentricity tolerance	CONC TOL
Flatness tolerance	FLAT TOL

Term	*Abbreviation*
Parallelism tolerance	PAR TOL
Positional tolerance	POSN TOL
Roundness tolerance	ROUND TOL
Straightness tolerance	STR TOL
Squareness tolerance	SQ TOL
Symmetry tolerance	SYM TOL
Tolerance zone (profiles)	TOL ZONE
Maximum metal, or material, condition	MMC
Full indicated movement	FIM

SECTION TWO : DIMENSIONING AND TOLERANCING

The following clauses relate mainly to drawings which define products in their completely finished state *as required by the designer*. Such drawings do not necessarily define the manufacturing methods by which the design requirements are met. Many of the principles and practices, however, can be applied to process drawings which may define products in a partly finished state.

11. GENERAL PRINCIPLES

a. Each dimension necessary for the complete definition of a finished product should be given on the drawing and should appear once only.

It should not be necessary for such a dimension to be deduced from other dimensions or for the drawing to be scaled.

NOTE. The term 'finished product' refers to the product in the condition in which it is to be used and includes any specified surface treatment or finish other than painting or lacquering.

A finished product drawing may define a piece ready for assembly or service ; or the product of a foundry, forge, etc.—supplied for further processing.

Any departure from the above principle should be limited to special cases and be the subject of an explanatory note.

b. There should be no more dimensions than are necessary to define the component, and no surface, line or point should be located by more than one toleranced dimension in any one direction.

NOTE. For exceptions to this general rule see Clause 13 *g*, page 38.

c. The functional dimensions* should be expressed directly on the drawing (see Fig. 38). The application of this principle will result in the selection of datum features on the basis of the function of the product and the method of locating it in any assembly of which it may form a part. If any datum feature other than one based on the function of the product is used, finer tolerances will be necessary and products which would satisfy the functional requirements may be rejected for exceeding these finer tolerances. (See Fig. 39.)

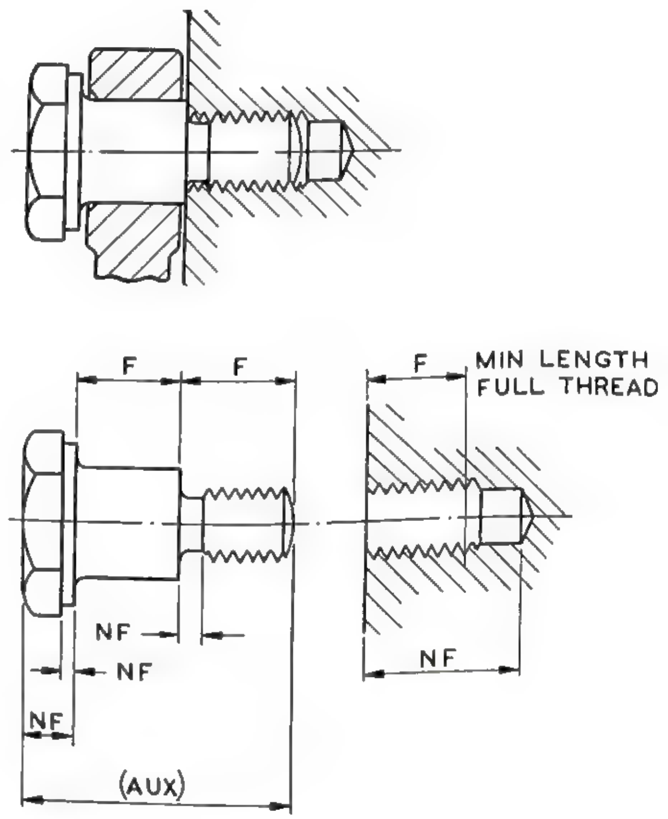
d. The non-functional dimensions should be placed in the manner most convenient to the producer or inspector.

e. Tolerances should be specified for all requirements affecting functioning or interchangeability wherever it is doubtful that ordinary or established workshop technique and equipment can be relied upon to achieve the required standard of accuracy. Tolerances should also be used to indicate where unusually wide variations are permissible.

f. Standard sizes should be used wherever practicable, particularly for drilled or reamed holes, thread sizes, nuts, bolts, studs, pins, etc., and for work which would be satisfactory with the sizes and surface finish of standard stock such as bright bar or extruded sections.

* A functional dimension is a dimension which directly affects the function of a product.

g. Production processes or inspection methods should not be specified unless they are essential to ensure satisfactory functioning or interchangeability. This does not apply to process drawings nor does it preclude the quoting of drill sizes.



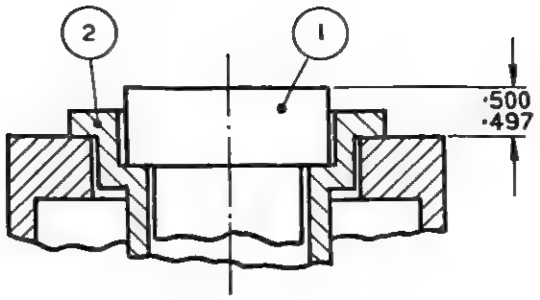
F = A functional dimension.

NF = A non-functional dimension.

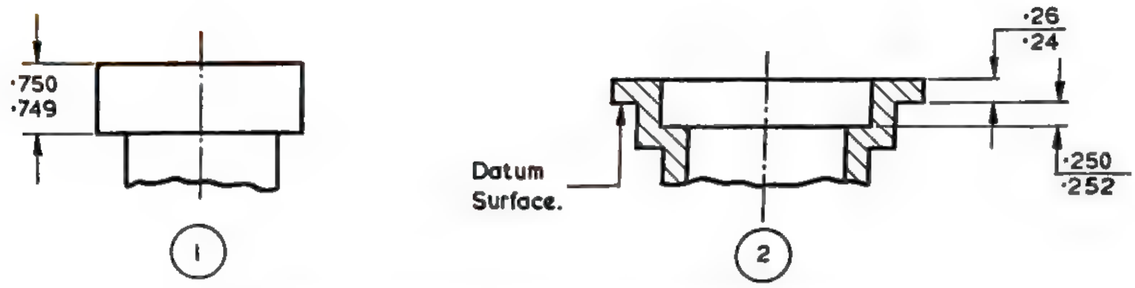
AUX = An auxiliary dimension given, without tolerances, for information only. (see clause 13 g, p. 38).

Fig. 38. Application of dimensions to suit design requirements

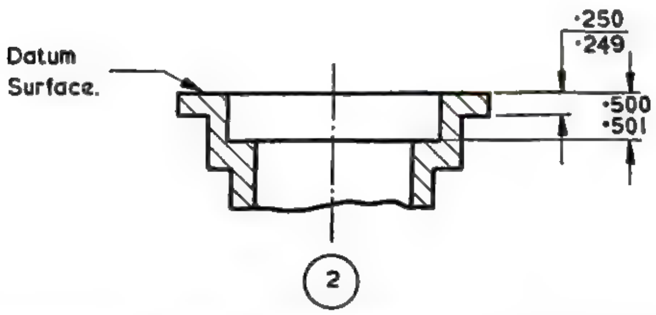
CLAUSE 11 (*cont.*)



(*a*) Assembly drawing showing functional requirement.



(*b*) Parts 1 and 2 dimensioned from functional datum surface.



(*c*) Part 2 redimensioned using top surface as datum. Tolerances have had to be reduced to keep assembly within the limits 0·500 and 0·497.

Fig. 39. Effect of changing datum surfaces from those determined by functional requirements

12. PROJECTION AND DIMENSION LINES AND LEADERS

a. Projection lines are thin full lines (Clause 5, type B, page 17) projected from points, lines or surfaces to enable the dimensions to be placed outside the outline wherever possible.

b. Where projection lines are extensions of lines of the outline, as in Fig. 40, they should preferably start just clear of the outline and should extend a little beyond the dimension line.

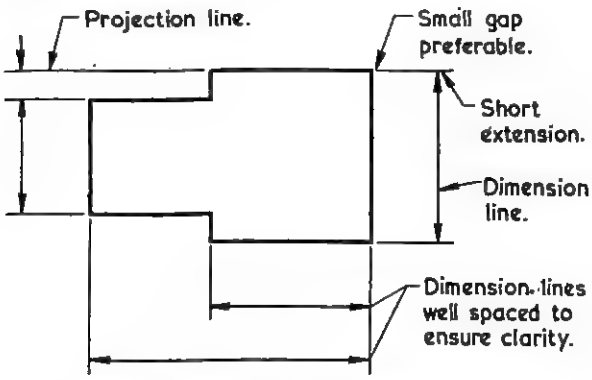


Fig. 40. Projection and dimension lines

c. Where projection lines refer to points on surfaces, or to points of intersection, they should touch or pass through the points, as shown in Figs. 41 and 42. To gain clarity the points may be emphasized by small dots, as in Fig. 42.

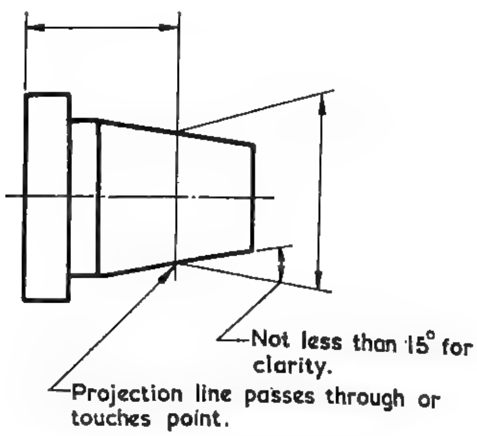


Fig. 41. Projection lines from points of intersection

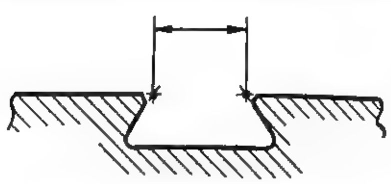
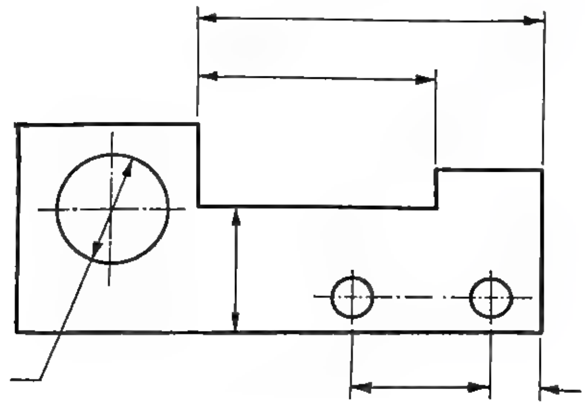


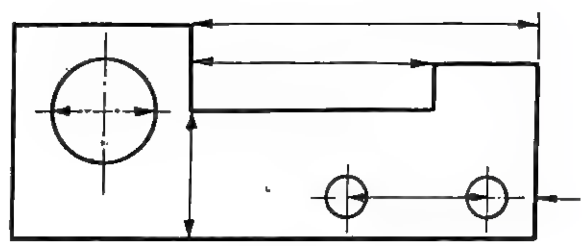
Fig 42. Points of intersection emphasised by dots

d. Dimension lines should be thin full lines (Clause 5, type B, page 17) and, wherever practicable, should be placed outside the outline of the object, as in Fig. 40. They may be interrupted for the insertion of dimensions. Arrowheads should be easily readable and normally not less than ⅛ in. long. It is important that the point should touch the projection or other limiting line.

e. A centre line, or a line which is an extension of a centre line or of part of an outline, should never be used as a dimension line (see Fig. 43).



(*a*) Correct.

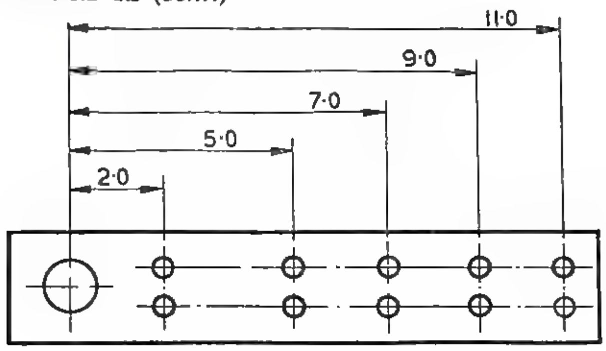


(*b*) Incorrect.

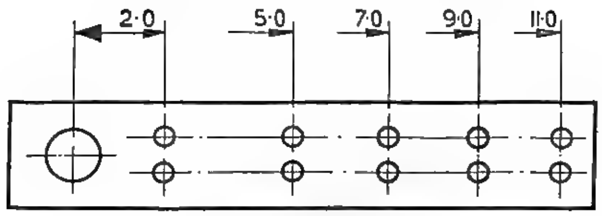
Fig. 43. Centre lines and extension lines must not be used as dimension lines

f. Where a number of dimensions are to be given from a common datum surface, line or point, one of the methods shown in Fig. 44 should be used. The normal method should be used wherever practicable. There are instances, however, where the alternative method has definite advantages, e.g. where space is restricted. Where the alternative method is used, the arrowhead on the datum line should be made approximately twice the normal size. In both methods it adds clarity to the drawing if the dimensions are placed near the appropriate arrowhead.

CLAUSE 12 (*cont.*)



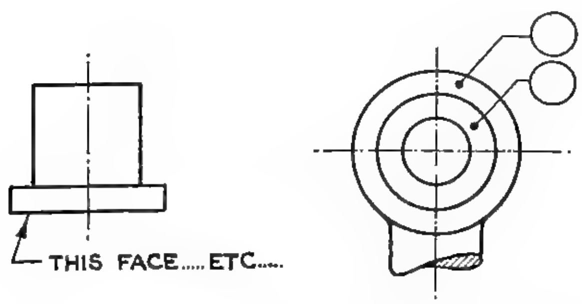
(*a*) Normal method.



(*b*) Alternative method.

Fig. 44. Dimensioning from a common datum

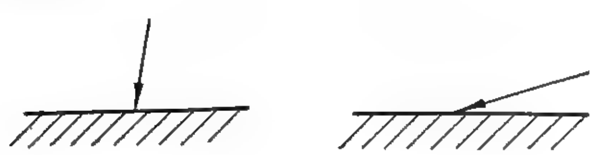
g. Leaders, used to indicate where dimensions or notes are intended to apply, should be thin full lines (Clause 5, type B, page 17) terminating in arrowheads or dots. Arrowheads should always terminate on a line ; dots should be within the outline of the object (see Fig. 45).



(*a*) Leader terminating in arrow head. (*b*) Leaders terminating in dots.

Fig. 45. Typical leaders

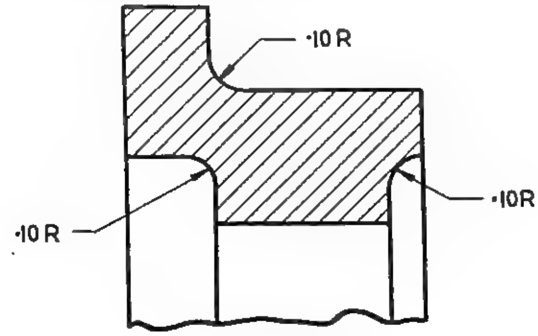
h. Leaders which touch lines should not do so at an acute angle (see Fig. 46) ; they should not be parallel to adjacent dimension or projection lines where confusion might arise.



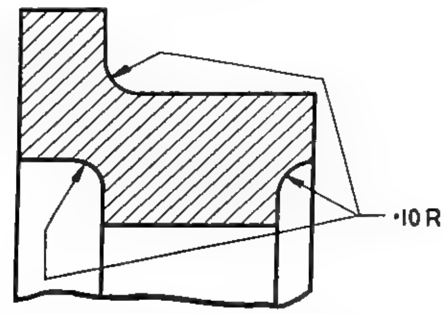
(*a*) Correct. (*b*) Incorrect.

Fig. 46. Leader lines touching other lines

j. The use of long leaders should be avoided even if it means repeating dimensions or notes (see Fig. 47) or using letter symbols (see Fig. 48).

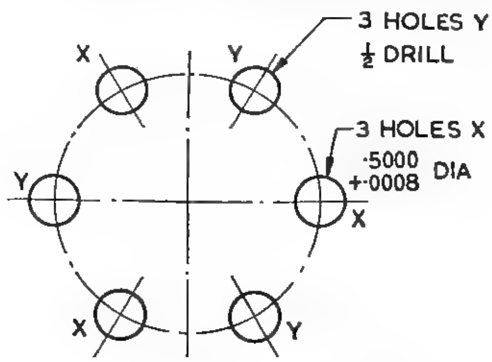


(*a*) Correct.

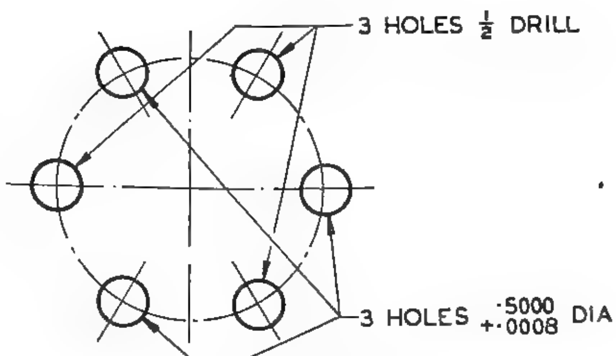


(*b*) Incorrect.

Fig. 47. Dimensions repeated to avoid long leaders



(*a*) Correct.



(*b*) Incorrect.

Fig. 48. Letter symbols to avoid long leaders

13. DIMENSIONS

a. *Units.*

(i) The standard method of dimensioning is in inches, or in feet and inches, expressed thus :

$7\frac{1}{2}''$; $7{\cdot}65''$; 6 FT. 2 IN. ; $5'-0\frac{1}{2}''$

(ii) Where the symbol ' is used to indicate feet, a long dash should be placed between the feet and inches. There is no need for this dash when the abbreviation 'ft' is used.

(iii) Fractional and decimal dimensions may be used on the same drawing (see also Clause 13 *c.* (iii)).

(iv) In general, dimensions up to and including 2 feet should be expressed in inches only. Above 2 feet, either inches only, or feet and inches, may be used.

(v) Where feet are used as one unit of a dimension, it is preferable to express parts of an inch in fractions, thus :

$5'-7\frac{1}{2}''$ and not $5'-7{\cdot}5''$

(vi) Where decimals are used, any figures preceding the decimal point should preferably be given in inches and not in feet and inches, thus :

$25{\cdot}42''$ and not $2'-1{\cdot}42''$

(vii) Where the metric system is used, dimensions should be expressed in millimetres.

(viii) Where the drawings are dimensioned mainly in one unit, i.e. mainly in inches or mainly in millimetres, the symbol " or the abbreviation 'mm' may be omitted, provided that a prominent note such as the following is placed on the drawing :

UNLESS OTHERWISE STATED, DIMENSIONS ARE IN INCHES.

b. *Fractions.*

(i) Fractions should preferably be shown with horizontal dividing lines, e.g. $\frac{1}{2}$, except where space is restricted when an oblique line may be used.

(ii) Fractions should be complete so as to avoid confusion with tolerances and should never be finer than $\frac{1}{64}$ in., e.g. :

$\frac{57}{64}$ and not $\frac{7}{8} + \frac{1}{64}$

c. *Decimals* (inches or millimetres)

(i) Decimal points should be bold, be given a full letter space, and be placed at the mid height of the numerals.
Where the dimension is less than unity, it is recommended that the decimal point is not preceded by the cipher '0', e.g. :

$\cdot 45$ and not $0{\cdot}45$

(ii) Decimal dimensions should be expressed to the least number of decimal places consistent with the design requirements and the use of standard basic sizes. While it would be unnecessary, in many cases, to express a dimension to more than two decimal places, such factors as the use of standard basic sizes based on a fractional inch series, or of stock materials, tools, etc., may necessitate the use of three or more decimal places.

(iii) Where drawings are dimensioned mainly in inch decimals, the sizes of holes produced by standard inch or metric drills, or of screw threads and other details known by their fractional inch or metric sizes, may be expressed in fractions of an inch or in millimetres. Where such sizes are expressed in inch decimals it is often helpful to give the parent dimension in brackets.
Where drawings are dimensioned mainly in millimetres the same principles should be applied to features which are generally known by their fractional inch sizes.

d. *Gauge numbers and drill numbers and letters.* Thicknesses or diameters indicated by gauge or drill numbers (or letters) should be accompanied by the size in decimals of an inch or millimetres as the case may be, thus :

10 SWG (·128); ·128 (10 SWG)

No 55 DRILL (·052); ·052 (No 55 DRILL)

F DRILL (·257); ·257 (F DRILL)

CLAUSE 13 (*cont.*)

e. Angles. Angular dimensions should be expressed in degrees, degrees and minutes, or in degrees, minutes, and seconds, e.g. :

$$22\tfrac{1}{2}^\circ$$
$$22^\circ\ 30'$$
$$52^\circ\ 30'\ 30''$$

Where an angle is less than one degree, it should be preceded by 0°, e.g. :

$$0^\circ\ 15'$$

f. Arrangement of dimensions.

(i) Dimensions should be placed so that they may be read either from the bottom, or the right-hand side, of the drawing (see Fig. 49).

(ii) Where there are several parallel dimension lines, the dimensions should be staggered as in Fig. 50 in order to avoid confusion.

(iii) Overall dimensions should be placed outside the intermediate dimensions as in Fig. 51.

(iv) Various methods of dimensioning narrow spaces are shown in Fig. 49.

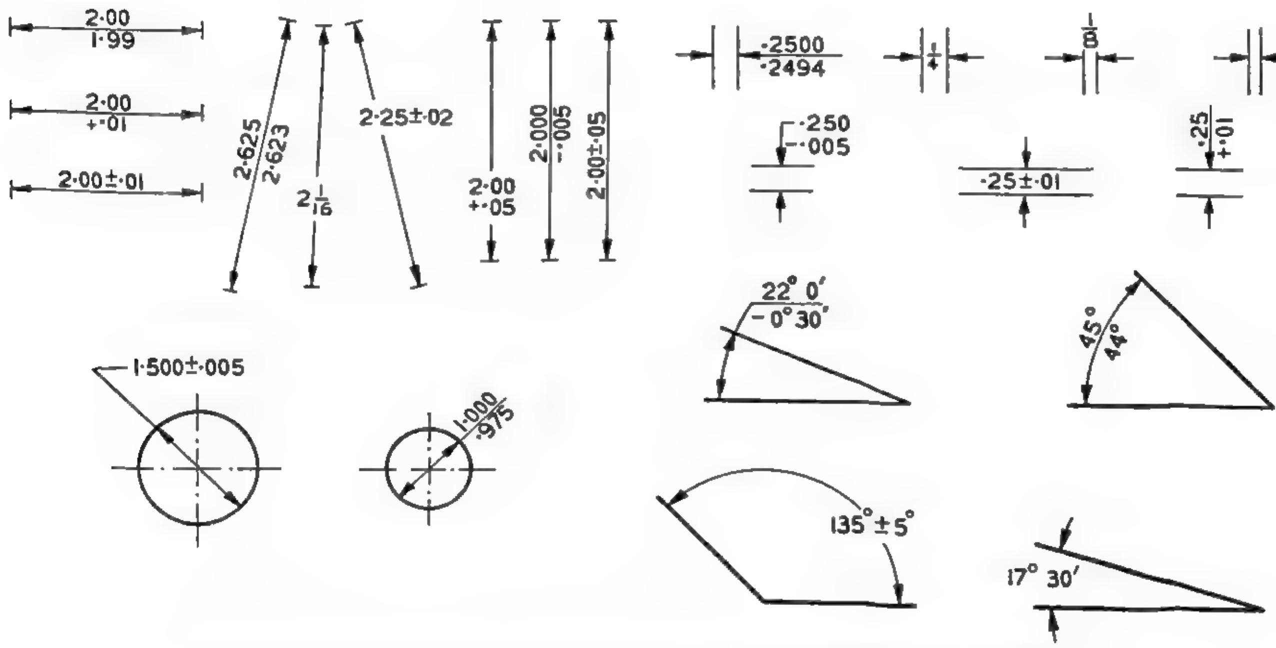


Fig. 49. Placing dimensions to read from the bottom or the right-hand side of the drawing

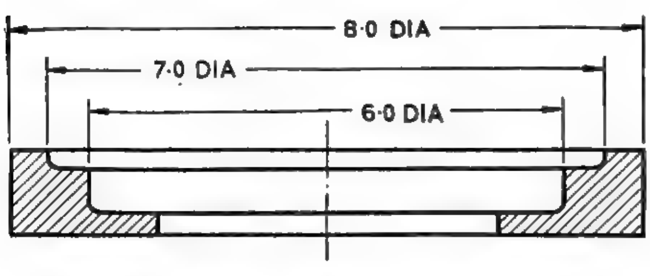


Fig. 50. Dimensions staggered to avoid confusion

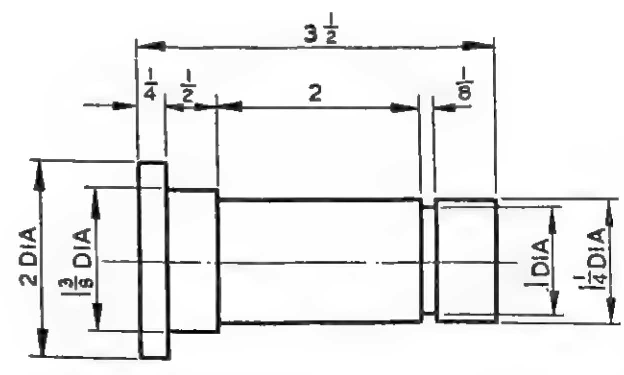


Fig. 51. Overall dimensions placed outside intermediate dimensions

CLAUSE 13 (*cont.*)

g. Redundant dimensions. Where an overall dimension is shown, one of the intermediate distances is redundant and should not be dimensioned (see Fig. 51). Exceptions may be made where redundant dimensions would provide useful information in which case they should be given as 'auxiliary' dimensions. Where all the intermediate dimensions are shown, the overall distance should generally be given as an auxiliary dimension (see Figs. 52 and 38).

Auxiliary dimensions should not be toleranced but should be included in brackets (see Figs. 52 and 53). Auxiliary dimensions relating to position should be based on the dimensions which define the true theoretical positions of the features concerned; where they relate to size, they should normally be based on the mean sizes of the features concerned. In other cases, the basis of calculation should be clearly stated on the drawing.

Auxiliary dimensions do not govern machining operations or acceptability in any way.

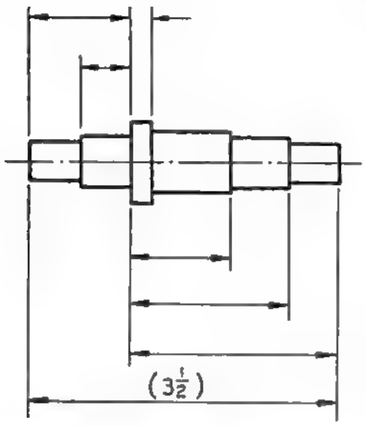


Fig. 52. Overall length added as an auxiliary dimension

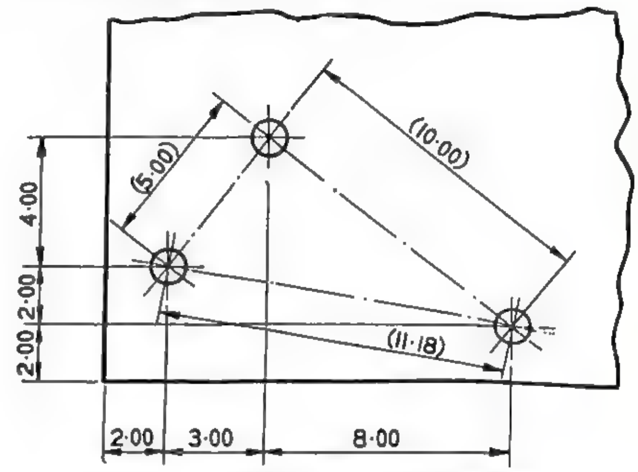


Fig. 53. Application of auxiliary dimensions

14. TOLERANCED DIMENSIONS

a. General. As it is impossible to manufacture components to exact sizes, it is necessary, in general, to introduce tolerances which are used to define the maximum and minimum 'limits of size' which can be accepted. The magnitude of such tolerances will vary widely according to circumstances; for example, in fine precision work the tolerance may necessarily be much less than 0·001 in., while a tolerance of 0·5 in. or more may be quite reasonable on some dimensions of a heavy casting.

Tolerances should be chosen with due regard to performance, possible difficulties in manufacture, inspection and assembly and should be as large as satisfactory functioning will permit. As a general rule, tolerances for fractional dimensions should be expressed as fractions, and for decimal dimensions as decimals.

b. Application of tolerances. Tolerances should be specified for all requirements critical to functioning and interchangeability wherever it is doubtful (as, for example, in most subcontracted work) that ordinary or established workshop technique and equipment can be relied upon to achieve a satisfactory standard of accuracy. Tolerances should also be specified to indicate where unusually wide variations are permissible. Tolerancing should be carried out *either* by a general note, or notes, assigning uniform or graded tolerances to specified classes of dimensions, *or* by tolerances assigned to individual dimensions.

(i) *General tolerances.* The use of general tolerance notes greatly simplifies the drawing and saves much labour in its preparation. The examples in Fig. 54 illustrate the wide field of application of this system.

TOLERANCE EXCEPT WHERE OTHERWISE STATED $\pm\cdot02$

TOLERANCES ON FRACTIONAL DIMENSIONS

6 IN. AND BELOW	$\pm\frac{1}{64}$
ABOVE 6 IN.	$\pm\frac{1}{32}$

TOLERANCE ON CAST THICKNESSES $\pm 12\frac{1}{2}\%$

TOLERANCES EXCEPT WHERE OTHERWISE STATED — ON DECIMAL DIMENSIONS

SIZE	TOLERANCE
UP TO 3·0	±·010
OVER 3·0 UP TO 12·0	±·015
OVER 12·0 UP TO 24·0	±·020
OVER 24·0 —	±·030

FOR TOLERANCES ON FORGING DIMENSIONS SEE B S 1718.

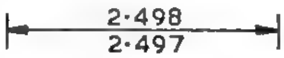
Fig. 54. Examples of general tolerance notes

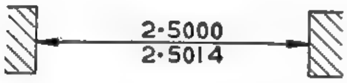
CLAUSE 14 (*cont.*)

(ii) *Individual tolerances.* Where it is required to tolerance an individual dimension, one of the following methods of expression should be used. There is no difference in the interpretation of these methods of expression, each of which does no more than define the maximum and minimum limits of size.

The associated dimensions and tolerances should always be expressed to the same number of decimal places and, where appropriate, the decimal points should be vertically in line.

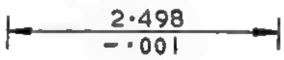
METHOD A: by specifying directly both limits of size, e.g.:

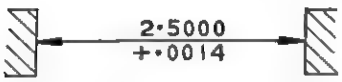




The high limit of size should be given first, except in the case of a *hole* or other internal feature where the *low* limit (maximum metal size) should be given first.*

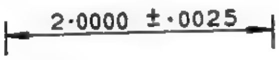
METHOD B: by specifying one limit of size with a tolerance in one direction only, e.g.:

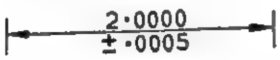




When using this method, the single limit of size expressed should normally be the maximum metal size. It follows that holes and internal features will normally carry a 'plus' tolerance, and shafts and external features a 'minus' tolerance.

METHOD C: by specifying a size with limits of tolerance above and below that size, preferably but not necessarily equally disposed; one of these limits of tolerance may be zero, e.g.:





$$\left.\begin{array}{l} 1{\cdot}2500\,\begin{smallmatrix}+{\cdot}0002\\-{\cdot}0001\end{smallmatrix} \\[1em] {\cdot}750\,\begin{smallmatrix}+0\\-{\cdot}002\end{smallmatrix}\end{array}\right\}\text{ Not recommended}$$

* Although the terms 'maximum metal' and 'minimum metal' have been used in the standard to denote the condition of most or least metal in the part respectively, they are intended to apply to parts manufactured from materials other than metal.

The following method (which should not be confused with method C) is sometimes found convenient in design offices *but is not recommended for use on drawings issued for purposes of manufacture.*

$$4{\cdot}500\,\begin{smallmatrix}+{\cdot}007\\+{\cdot}005\end{smallmatrix}$$

$$3{\cdot}250\,\begin{smallmatrix}-{\cdot}003\\-{\cdot}004\end{smallmatrix}$$

c. Single limits of size. Where it is necessary to specify only one limit of size of a dimension (e.g. the minimum length of full thread or the maximum radius that is permitted in a corner), the abbreviation 'MAX' or 'MIN' should be used, e.g.:

·75 MIN LENGTH FULL THREAD

·02 R MAX

d. Angular dimensions. Angular dimensions and their tolerances should contain the same units of measurement, as in the following examples:

35° 34°	30° 30′ 30° 0′
22° –1°	47° 0′ 0″ –0° 0′ 30″
90° ±2°	0° 30′ 0″ ±0° 1′ 30″

A full space should be left between the degree symbol and the minute figure, and between the minute symbol and the seconds figure.

CLAUSE 14 (*cont.*)

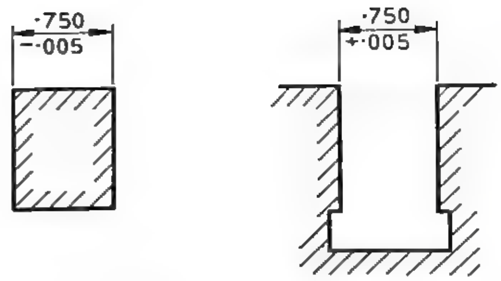
e. Interpretation of limits of size of features. Toleranced dimensions for the sizes of features (however expressed) control form as well as size.

In theory, the maximum metal limit of size (i.e., the high limit of size of an external feature or the low limit of size of an internal feature) defines a limit of perfect form for the relevant surfaces. In other words, if a feature is everywhere on its maximum metal limits of size, it should be perfect in form. Errors of form may occur provided no part of the finished surfaces crosses the maximum metal limit of form, and the feature is in accordance with its specified limits of size. Fig. 55 shows, diagrammatically, typical extreme errors of form which could be accepted by this interpretation. However, whilst the extreme errors shown are possible, they are unlikely to arise in ordinary machining practice. This control is identical with that exercised by ordinary limit gauges where the faces of the GO gauges are large enough to cover the entire work surfaces.

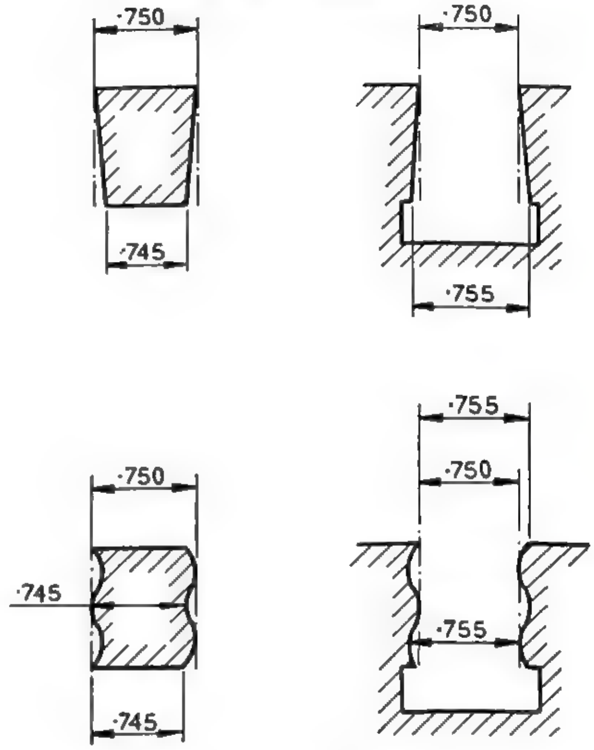
In practice, however, the relative proportions of the gauging faces of ordinary GO gauges and the surfaces of work features are usually such that these limit gauges cannot prove that the entire work surfaces are within the maximum metal limit of form. Nevertheless, the control imposed by such gauges on machined work is ordinarily regarded as adequate.

In those cases where it is important that the form of a feature, if it is in its maximum metal condition, must be exact and it is doubtful whether ordinary manufacturing technique and equipment will ensure sufficient accuracy, the letters 'MMC' (maximum metal condition) should be used as in Fig. 56 (*a*). Without this qualification there is no guarantee of accuracy of form. Fig. 56 (*b*) shows the requirements imposed by tolerancing a dimension in conjunction with such a note, the dotted lines representing the maximum metal limits of form. The finished surfaces should not lie outside these limits and the size across the flats should nowhere be less than 0·74.

Other methods of controlling form are dealt with in subsequent clauses.



(*a*) Drawing specifications.



(*b*) Possible extreme errors of form permitted by ordinary limit gauges when work is on minimum metal limit.

Fig. 55. Interpretation of limits of size by ordinary limit gauging practice. The finished surfaces must not cross the planes defining the maximum metal limits of form (shown by chain lines)

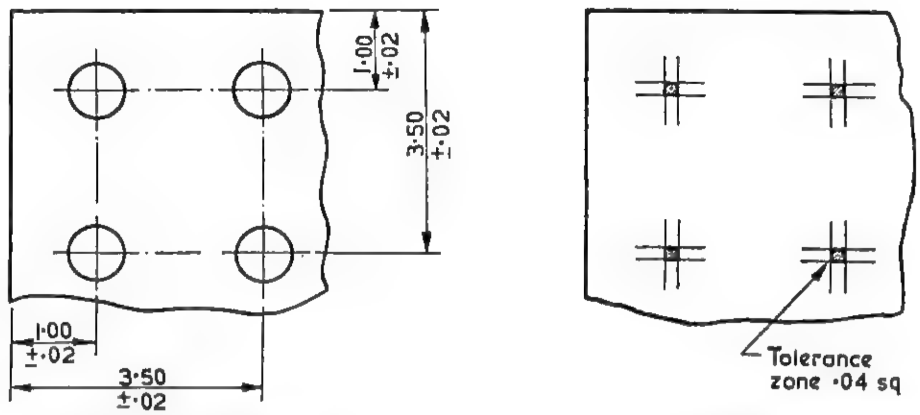
(*a*) Drawing specification. (*b*) One interpretation of (*a*).

Fig. 56. Type of note used to specify that the form is to be correct if the feature is in its maximum metal condition

CLAUSE 14 (*cont.*)

f. Interpretation of toleranced centre distances. Limits of centre distances may be expressed by any one of the methods A, B or C. The interpretation of toleranced centre distances is shown in Fig. 57 (*a*) and (*b*).

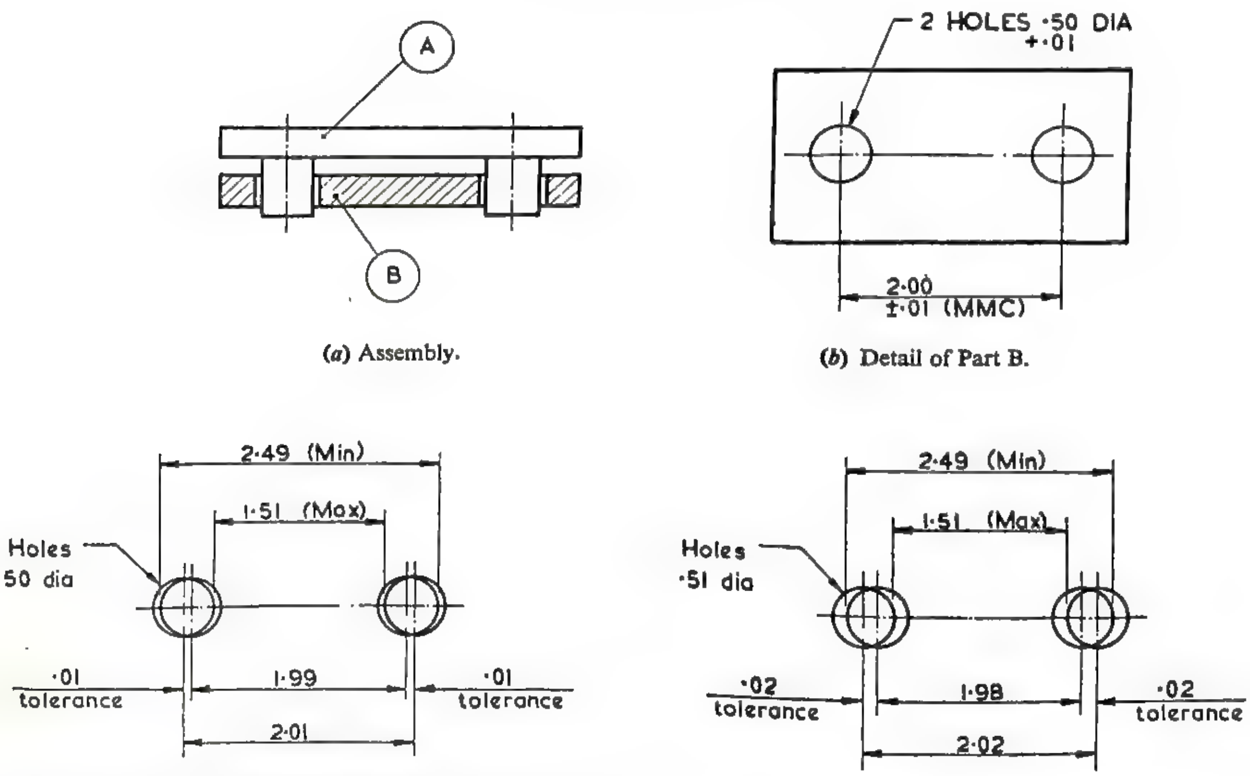
Except where otherwise indicated, the limits of centre distances must be observed regardless of the actual finished sizes of the features concerned. However, in many applications, e.g. Fig. 58 (*a*), the limits of centre distance may be exceeded if the features are not on their



(*a*) Drawing specification.

(*b*) Interpretation of (*a*)

Fig. 57. Interpretation of toleranced centre distances



(*a*) Assembly.

(*b*) Detail of Part B.

(*c*) Tolerance diagram for maximum metal condition.

(*d*) Tolerance diagram for minimum metal condition.

Fig. 58. Interpretation of toleranced centre distances specified in relation to maximum metal condition

CLAUSE 14 (*cont.*)

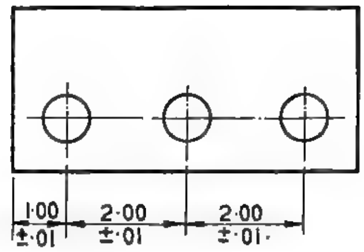
maximum metal limits of size, because of the increased clearances between the mating features. In such cases the toleranced dimensions of centre distance should be marked with the letters ' MMC ' (maximum metal condition), as in Fig. 58 (*b*), to indicate that the limits of centre distance must be observed if the features are on their maximum metal limits, but may be increased when the features are finished away from their maximum metal limits in the direction of minimum metal. The centre distances could then be checked by a simple gauge similar to part A in Fig. 58 (*a*) instead of by direct measurement.

In cases where toleranced centre distances are used it may be necessary to minimise the cumulative effects of the tolerances. Progressive dimensioning from a common datum is one method of reducing this accumulation.

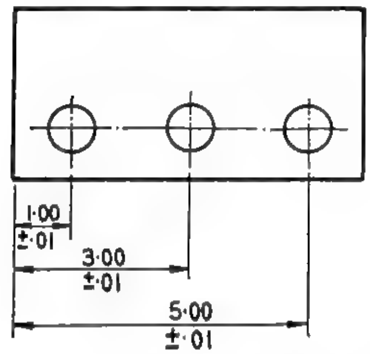
Fig. 59 compares chain and progressive dimensioning. It will be seen that, in this case, progressive dimensioning avoids the accumulation of tolerances in relation to the edge of the plate although, incidentally, the pitch variation is not the same in both cases.

Toleranced centre distances are suitable for defining the distance between two features (e.g. for the position of a hole relative to a flat surface or the distance between a pair of holes) particularly where the magnitude of the tolerance is different in two directions. Some typical applications of toleranced centre distances are shown in Fig. 60.

NOTE. It should be noted that toleranced centre distances are normally checked individually, i.e. from feature to feature. Therefore, where there are more than two features which need to be related together as a group, the use of positional tolerances should be considered because they avoid accumulation of tolerances and enable the requirements to be specified more precisely (see Clause 19 *e*, page 66).



(*a*) Chain dimensioning may result in an accumulation of tolerance between edge of plate and second and third holes.



(*b*) Progressive dimensioning avoids the accumulation of tolerance between edge of plate and second and third holes.

Fig. 59. Comparison of chain and progressive methods of dimensioning using toleranced centre distances

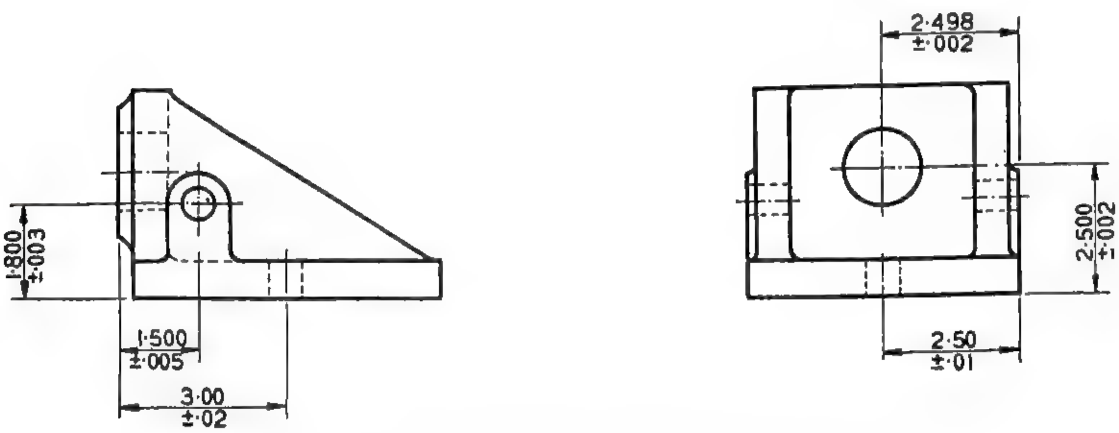


Fig. 60. Dimensioning positions by toleranced centre distances

15. NOTES

Notes should not be underlined except where special emphasis is required. They may be classified as General or Local.

a. General notes. General notes may be used with advantage to specify requirements which would otherwise need to be repeated many times. Typical examples of general notes are given below.

DIMENSIONS ARE IN INCHES UNLESS OTHERWISE STATED.

SHARP EDGES TO BE REMOVED.

CASTING (OR FORGING ETC) RADII ARE $\frac{1}{4}$ UNLESS OTHERWISE SPECIFIED.

FOR ACCEPTANCE TESTS SEE SPEC No 432

b. Local notes. These refer to local requirements and should be placed near the point to which they refer. Fig. 61 is an example of a local note.

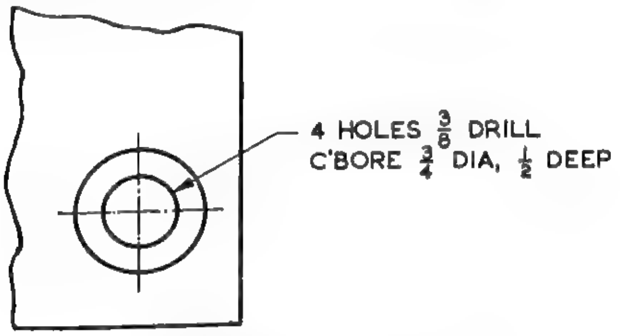


Fig. 61. Example of local note

c. Method of indicating special inspection requirements. Where it is required to draw the particular attention of the Inspector to any specific requirement on the drawing, that requirement should be marked with the symbol ⊗ and the following note added to the general notes on the drawing :—

⊗ THIS SYMBOL IS FOR INSPECTION REFERENCE ONLY.

The use of this symbol implies that the inspector will receive special instructions regarding the requirement so marked.

Examples of the application of this symbol are shown below :—

⊗ FLAT TOL ·001 WIDE.

⊗ CLEAN BY PROCESS 'B' BEFORE SOLDER SEALING.

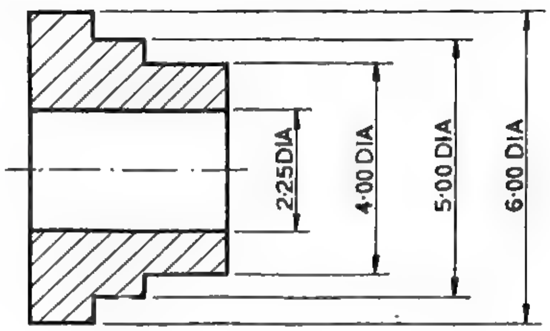
WEIGHT OF PULL-OFF 6 LB MAX ⊗
$4\frac{1}{2}$ LB MIN ⊗

HARDEN AND TEMPER, DPN 550 MIN ⊗
DPN 650 MAX

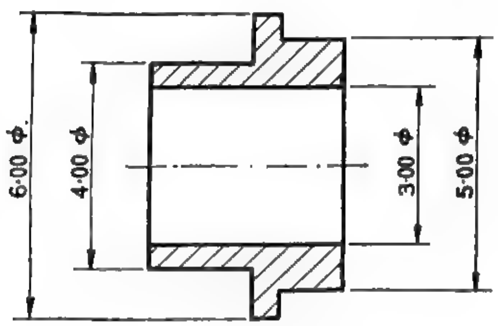
16. METHODS OF DIMENSIONING COMMON FEATURES

a. Diameters.

(i) A dimension indicating the diameter of a circle or cylinder should be followed by the abbreviation 'DIA' or symbol ϕ (see Fig. 62). Where it is obvious from the drawing that the dimension is a diameter the abbreviation or symbol may be omitted.



(*a*)



(*b*)

Fig. 62. Use of abbreviation 'DIA' and symbol ϕ

(ii) Dimensions of diameters should be placed on the most appropriate view to ensure clarity, as for instance, on a longitudinal view in preference to an end view consisting of a number of concentric circles (see Fig. 63).

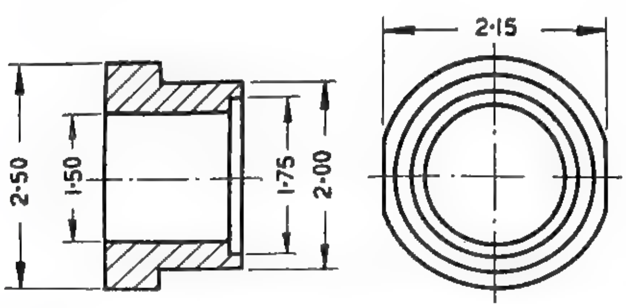


Fig. 63. Dimensions of diameters placed on most appropriate view for clarity

(iii) Dimension lines may sometimes be omitted and the dimensions related to the features by leaders as in Fig. 64.

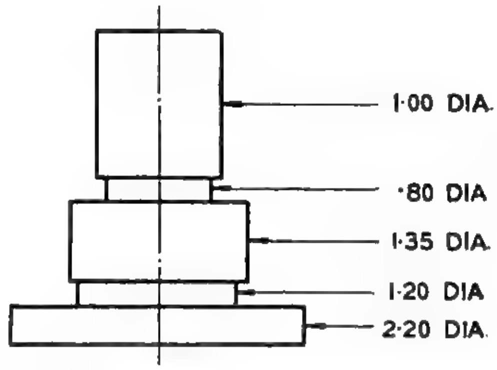
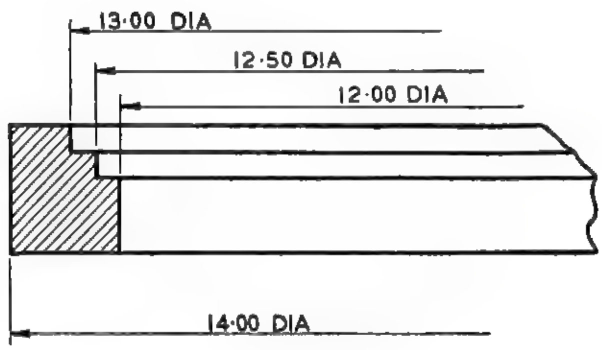
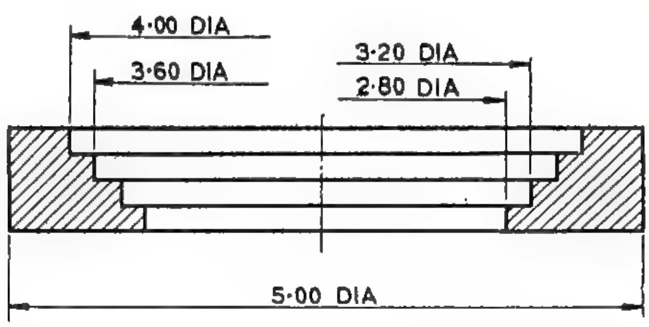


Fig. 64. Dimensions related to features by leaders

(iv) Where space is restricted, one of the methods shown in Fig. 65 may be used.



(*a*)



(*b*)

Fig. 65. Dimensioning diameters where space is restricted

CLAUSE 16 (*cont.*)

(v) Circles should be dimensioned by one of the methods illustrated in Fig. 66.

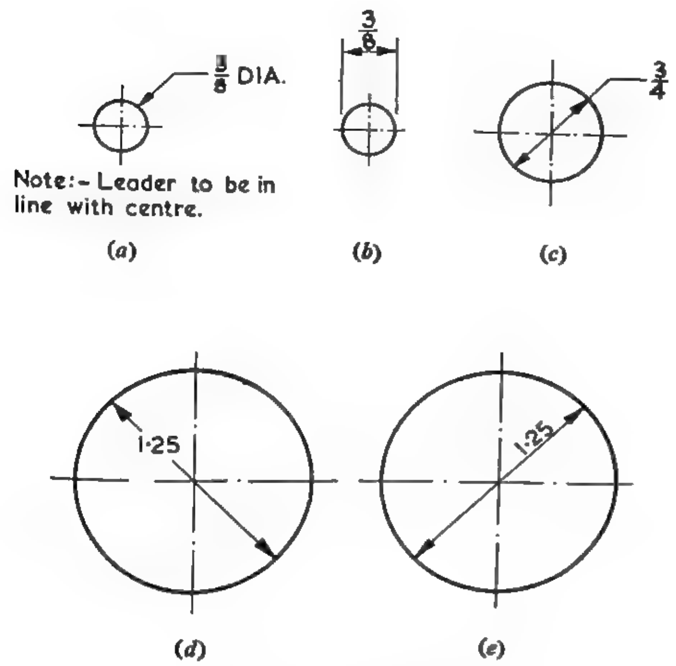


Fig. 66. Dimensioning circles

(vi) The diameter of a spherical surface should be dimensioned as in Fig. 67.

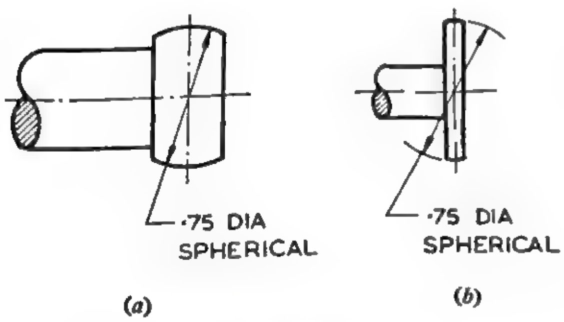


Fig. 67. Spherical diameters

b. Radii.

(i) Radii should be dimensioned by a dimension line which passes through, or is in line with, the centre of the arc. The dimension line should have one arrow-head only, that touching the arc. The abbreviation ' R ' or ' RAD ' should always follow the dimension.

(ii) Radii of arcs which need not have their centres located should be dimensioned by one of the methods shown in Fig. 68.

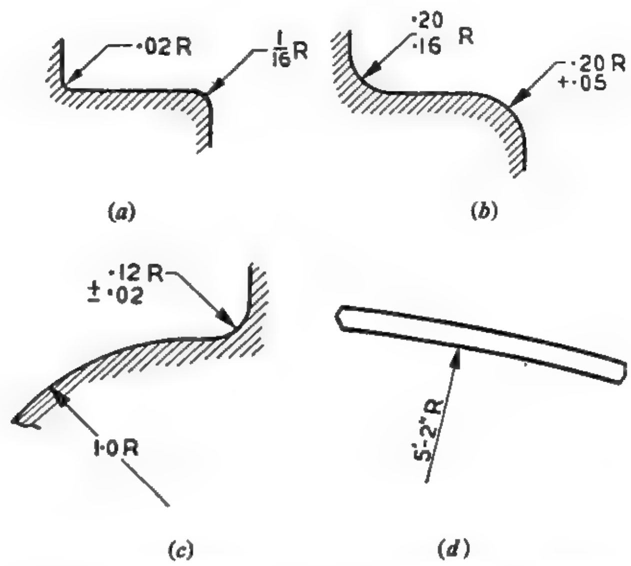


Fig. 68. Dimensioning radii of arcs which need not have their centres located

(iii) Where the centre of an arc cannot conveniently be shown in its correct position, and yet needs to be located, one of the methods illustrated in Fig. 69 should be used. The portion of the dimension line which touches the arc should be in line with the true centre.

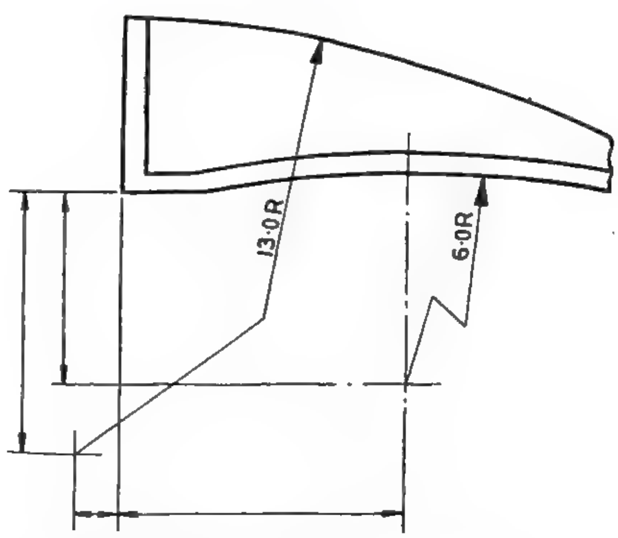


Fig. 69. Locating inconveniently placed centres

CLAUSE 16 (*cont.*)

(iv) The radius of a spherical surface should be dimensioned as in Fig. 70.

c. Holes : sizes. Typical methods of dimensioning holes are shown in Fig. 71. Suitable methods of production (e.g. drill, punch, ream, core, etc.) may be specified where appropriate. The depth of drilled holes, when stated in note form, refers to the depth of the cylindrical hole and not to the point of the drill.

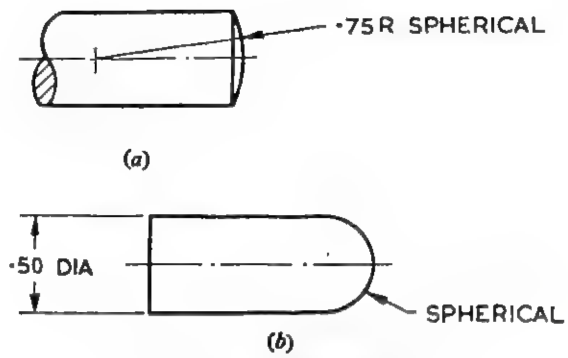


Fig. 70. Spherical radii

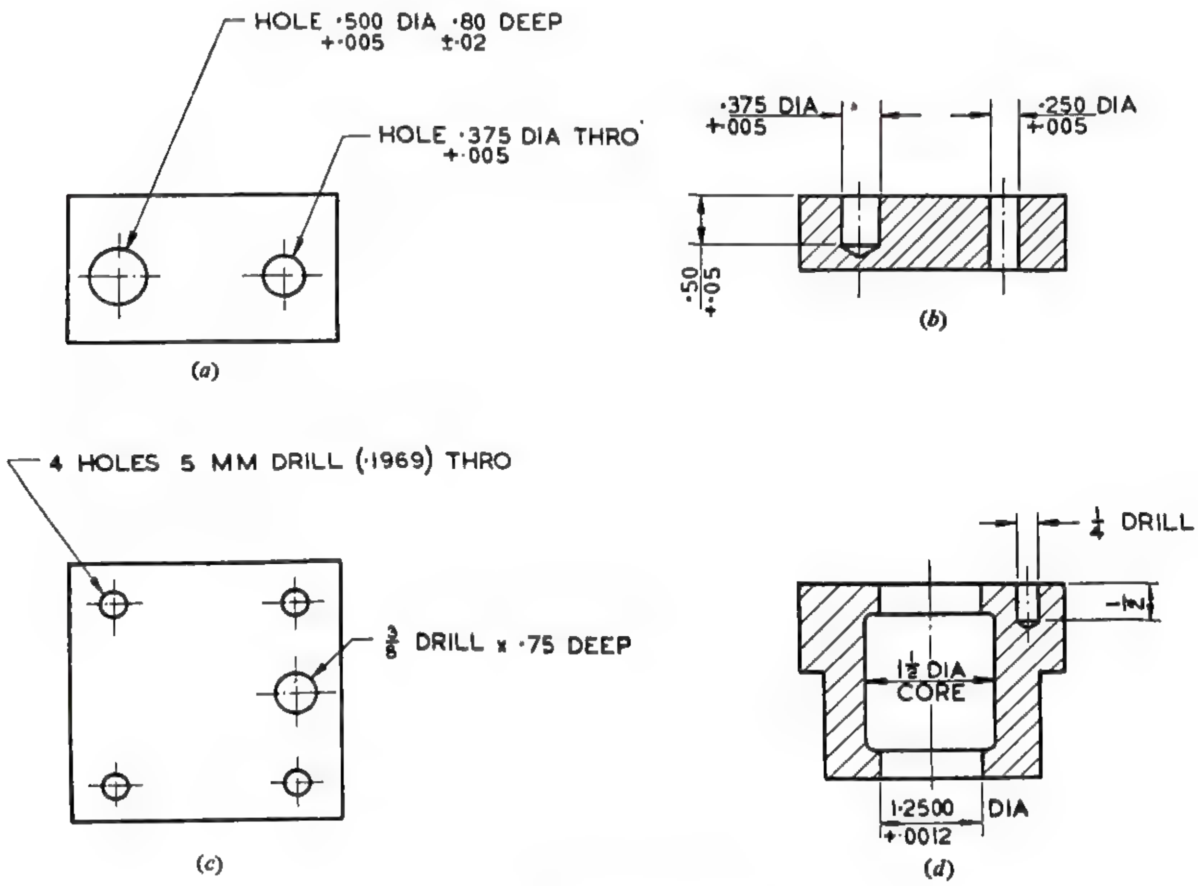
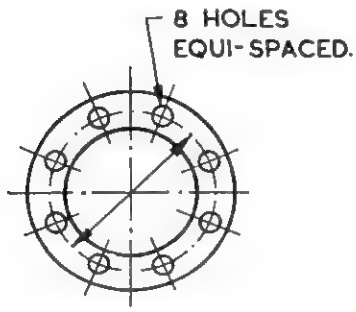


Fig. 71. Dimensioning holes

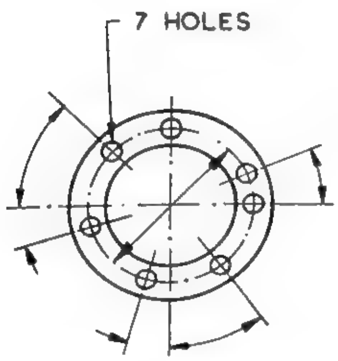
CLAUSE 16 (*cont.*)

d. Positioning holes and other features.

(i) The positions of holes and other features should be defined by spacing them on pitch circles as shown in Fig. 72, or by giving the rectangular co-ordinates or centre distances as shown in Fig. 73.



(*a*) Holes equally spaced.

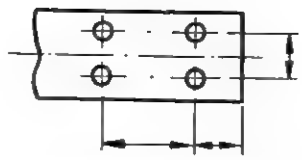


(*b*) Holes unequally spaced. Alternatively several or all of the angular dimensions may be given from one centre line.

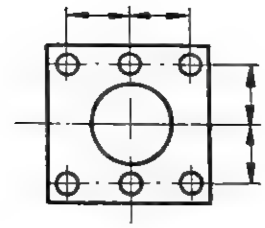
Fig. 72. Dimensioning positions of holes by angular spacings on a pitch circle.

(ii) In Fig. 72, the co-ordinates, and in Fig. 73 (*e*) the pitch circle radius and chordal pitches, may be added to the drawing as auxiliary dimensions (Clause 13 *g*, page 38).

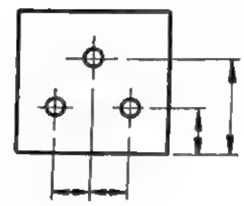
(iii) Features which are drawn, but not dimensioned, about a common centre line, as shown in Fig. 73 (*a*), are intended to be equi-spaced about that centre line.



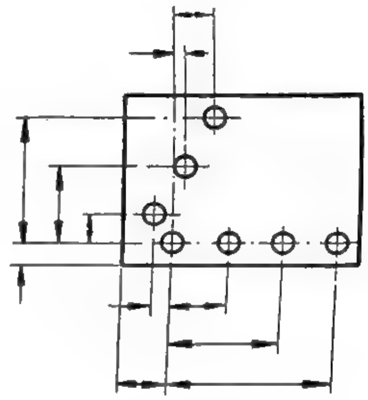
(*a*) Symmetry of holes about centre line implied



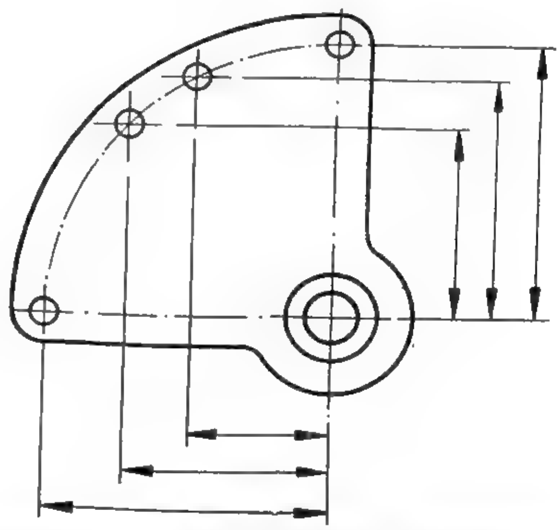
(*b*) Symmetry of holes about centre line defined



(*c*) Relation to lower edge important.



(*d*) Use of ordinates in preference to angular dimensions.



(*e*) Use of ordinates for holes lying on a pitch circle.

Fig. 73. Dimensioning positions of holes by ordinates

CLAUSE 16 (*cont.*)

e. Dimensioning on curved surfaces. In dimensioning the spacing of holes and other features on a curved surface, care should be taken to indicate clearly the surface on which the dimensioned points are to be measured and whether the dimensions are chordal or circumferential (see Fig. 74).

f. Countersinks, counterbores and spotfaces. When dimensioning a countersink, counterbore or spotface, the required dimensions should be given as shown in Fig. 75. Such notes as 'Counterbore to suit ¼ B.S. Fine Ch. hd. screw' should not be used.

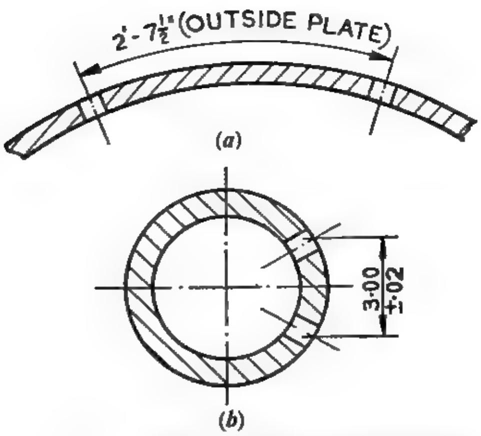


Fig. 74. Dimensioning on curved surfaces

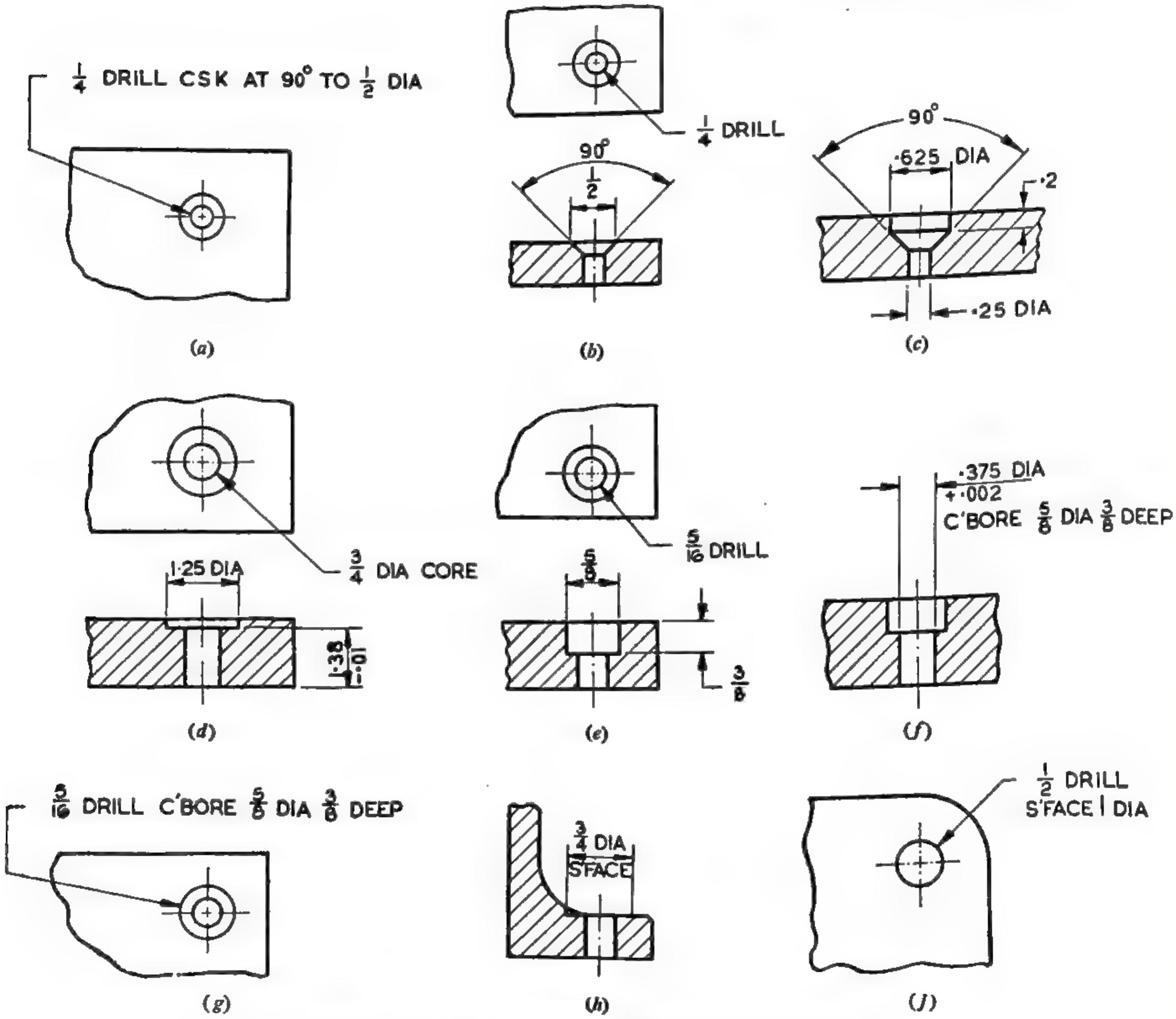


Fig. 75. Dimensioning countersinks, counterbores and spotfaces

CLAUSE 16 (*cont.*)

g. Chamfers. To avoid any misinterpretation of the dimensions of 45° chamfers, they should be specified by one of the methods shown in Fig. 76 and not by a note and a leader.

Chamfers at angles other than 45° should be dimensioned as shown in Fig. 77.

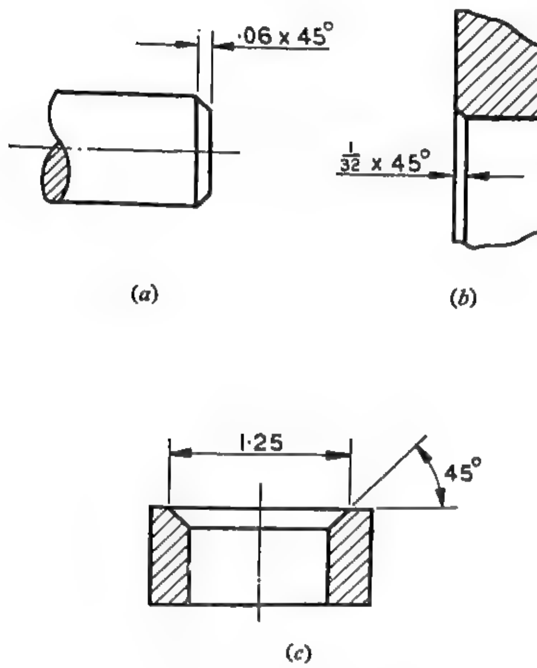


Fig. 76. Chamfers at 45°

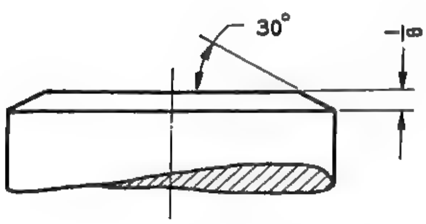


Fig. 77. For chamfers at angles other than 45°, the angle should be shown as a dimension and not as a note

h. Screw threads.

(i) *Designation.* Screw threads should be specified by using the designations recommended in the appropriate British Standard.

When specifying special screw threads, *the tolerances of which need to be calculated,* the dimensions for the major, effective and minor diameters should be given as in Fig. 78 or Plate 1.

(ii) *Undercut.* Where an undercut is necessary, it should be dimensioned on the drawing in accordance with the recommendations of B.S. 1936, 'Undercuts and runouts for screw threads'.

(iii) *Length of thread (parallel threads).* The length of full thread*, or the distance to the end of full thread, should be specified where necessary, using one of the methods shown in Figs. 79, 80 and 81.

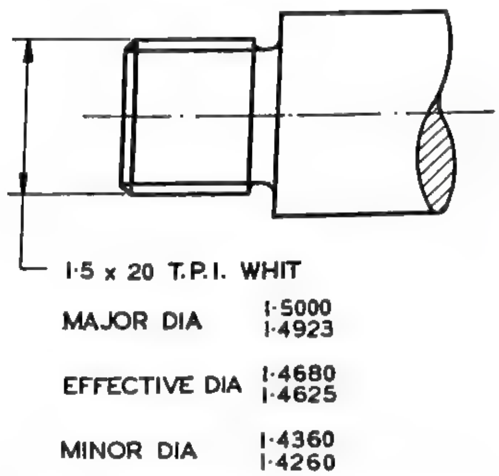


Fig. 78. Dimensioning a special screw thread

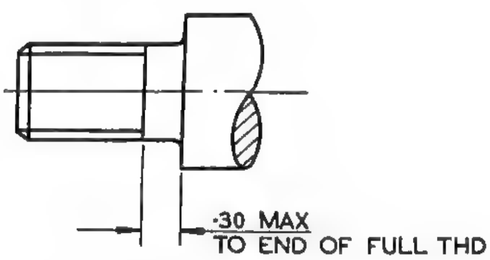


Fig. 79. Dimensioning to end of full thread

* The end of full thread is the point at which the root ceases to be fully formed. The root diameter of an external thread is the minor diameter and that of an internal thread the major diameter.

CLAUSE 16 (*cont.*)

Where it is necessary to limit the lengths of full and imperfect threads the method shown in Figs. 80 and 81 (*d*) should be used. In deciding those dimensions, reference should be made to B.S. 1936, 'Undercuts and runouts for screw threads'.

(iv) *Threaded holes.* Threaded holes should be dimensioned by one of the methods shown in Fig. 81.

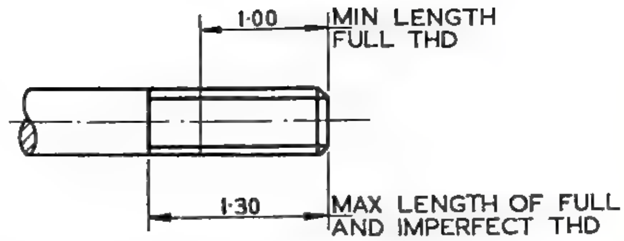


Fig. 80. Dimensioning to ends of full and of imperfect thread

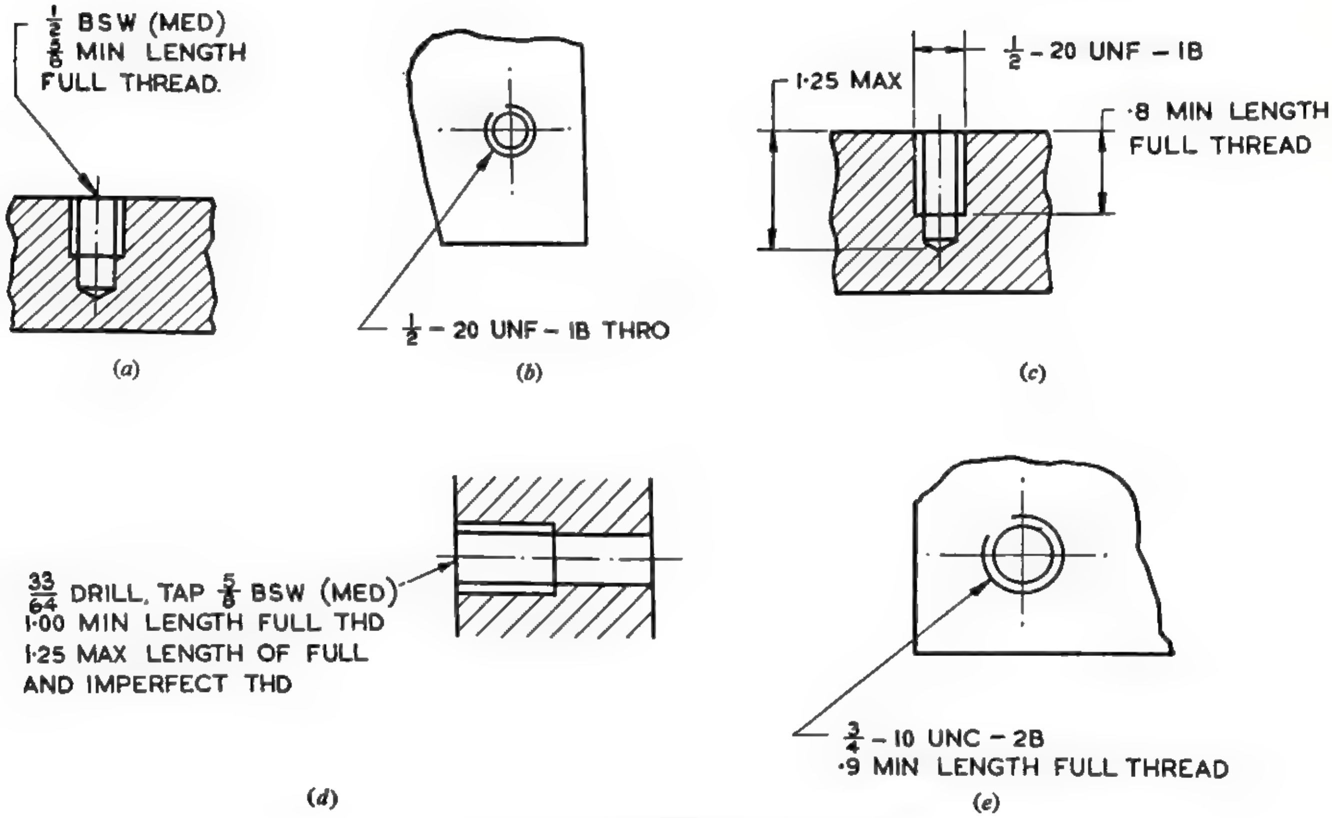
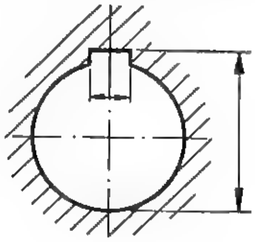


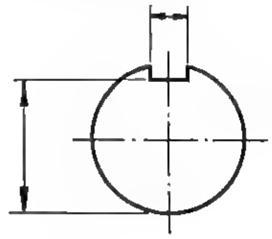
Fig. 81. Dimensioning threaded holes

17. KEYWAYS

The preferred methods of dimensioning keyways in hubs and shafts are shown in Figs. 82 and 83.

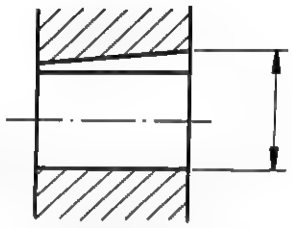


(*a*) Hub



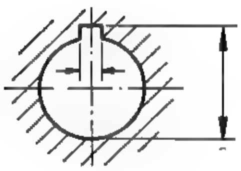
(*b*) Shaft

For keyways for tapered keys the depth of the keyway in the hub should be given at the deep end (c)

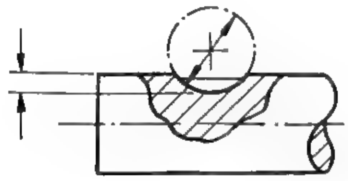


(*c*) Tapered keyway in hub

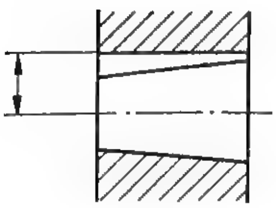
Fig. 82. Keyways for square and rectangular keys



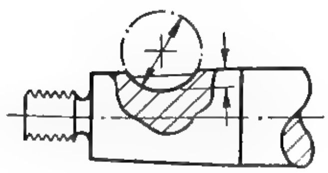
(*a*) Parallel hub



(*b*) Parallel shaft



(*c*) Tapered hub



(*d*) Tapered shaft

Fig. 83 Keyways for woodruff keys

18. TAPERED FEATURES

a. *Dimensioning.* The following dimensions may be used, in suitable combinations, to define the size and form of tapered features :

The diameter (or width) at each end of the tapered feature.

The length of the tapered feature.

The diameter (or width) at a selected cross-sectional plane which may be within the tapered feature or outside.

The dimension locating a cross-sectional plane at which a diameter or width is specified.

The rate of taper, or the included angle.

No more of these dimensions than are necessary should be given. However, additional dimensions, required for information purposes, may be given as auxiliary dimensions, (Clause 13 *g*, page 38). Fig. 84 shows some typical combinations of dimensions for specifying the size and form of tapered features.

b. *Tolerancing.* There are three methods of specifying the required accuracy of tapered features :

The basic taper (or angle) method, where the accuracy of the rate of taper is controlled solely by a tolerance on size.

The toleranced taper (or angle) method, where the angle or rate of taper is directly toleranced independently of the tolerance on size.

Fitting to gauge or mating part.

c. *The basic taper or angle method.* The term ' basic taper ' or ' basic angle ' means that the tolerance specified for the size of the feature applies at all cross-sectional planes throughout its length and so limits errors of form as well as errors of size.

Fig. 85 (*a*) shows a tapered feature dimensioned by a basic taper and with its size specified by a toleranced dimension at one end. The tolerance diagram in Fig. 85 (*b*) illustrates how the tolerance of 0·002 applies at all cross-sectional planes throughout the length of the tapered feature.

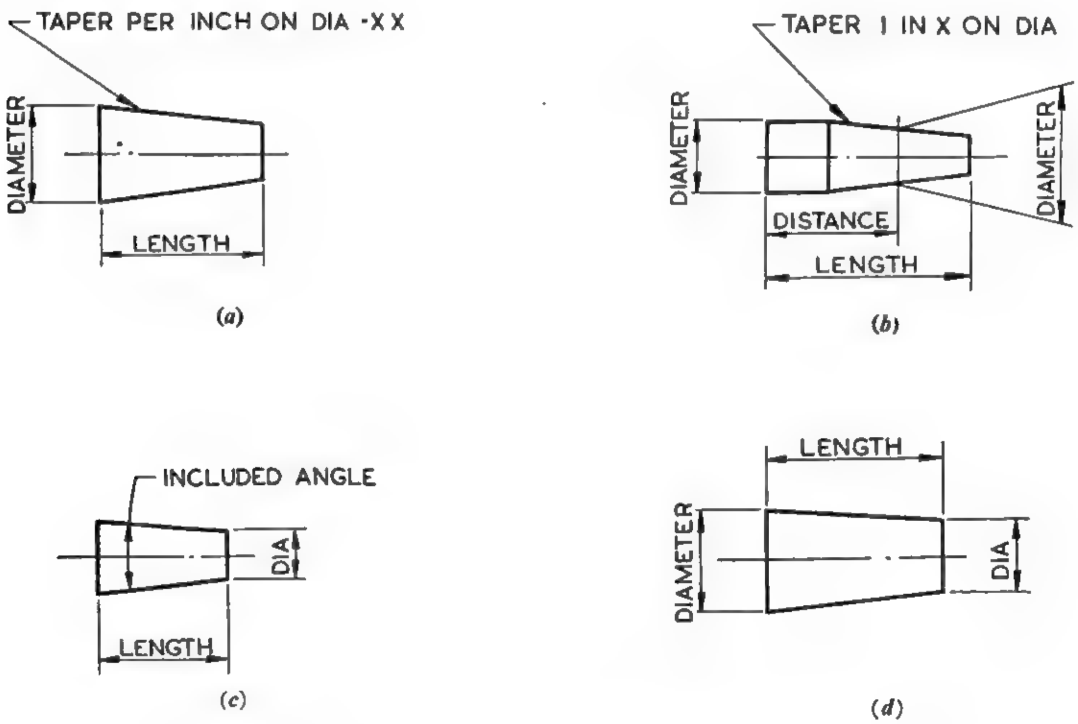
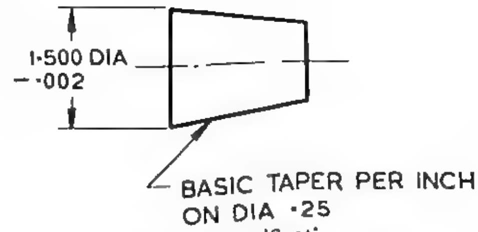
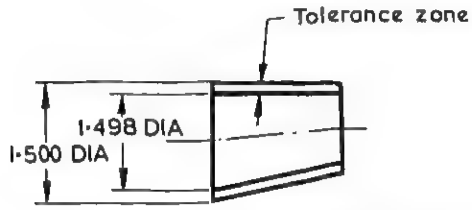


Fig. 84. Methods of dimensioning tapered objects

CLAUSE 18 (*cont.*)



(*a*) Drawing specification.



(*b*) Tolerance diagram for (*a*). The finished conical surface must lie inside the tolerance zone.

Fig. 85. Tolerancing a tapered object by the basic taper (or angle) method

Fig. 86 (*a*) shows a tapered feature dimensioned by a basic taper and with its size specified by a toleranced dimension at a plane located by a datum dimension.* The tolerance diagram in Fig. 86 (*b*) illustrates how the tolerance of 0·002 applies at all cross-sectional planes throughout the length of the tapered feature.

Fig. 87 (*a*) illustrates the use of a basic taper in conjunction with a datum dimension* which defines a cross-sectional plane which must be located within specified limits in relation to the left end of the piece. Fig. 87 (*b*) gives the tolerance diagram that results from the application of the 0·004 tolerance to the location of all cross-sectional planes throughout the length of the tapered feature.

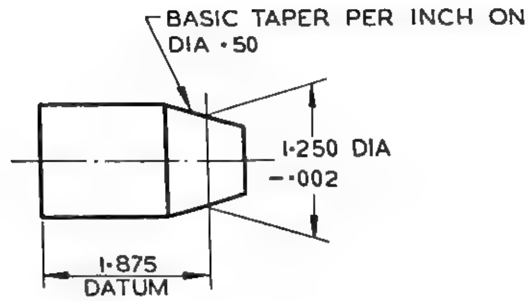
The tolerance diagrams (Figs. 86 (*b*) and 87 (*b*)) show that the nature of the control of size, form and location is the same whenever a basic taper (or angle) is specified.

It should be noted that, where the method of dimensioning shown in Figs. 86 (*a*) or 87 (*a*) is used, either the diameter, or the distance, must be a datum dimension.* If both were directly toleranced, the tolerances would be cumulative in their effect on the location of the conical surface in relation to the end datum face.

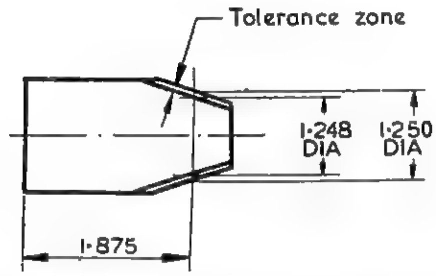
NOTE 1. It is not intended that the tolerance diagrams, Figs. 85 (*b*), 86 (*b*), etc., should appear on detail drawings; they are included here only to illustrate the interpretation.

NOTE 2. For simplicity, the tolerance diagrams in Figs. 85, 86, etc. show the minimum metal outlines symmetrically disposed with respect to the maximum metal outlines. In practice, this will not be far from the truth, although there is, in fact, no minimum metal limit of perfect form. Within the maximum metal outline any error of form may be present, providing the feature is everywhere within its minimum metal limits of size (see Clause 14 *e*, page 40).

* A datum dimension is a theoretically exact dimension which locates a datum point, line or plane at which a feature must be within certain limits of size, or to which other features are dimensioned for position, etc.; it is also used to define a cross-sectional plane of a feature the location of which may vary within specified limits.

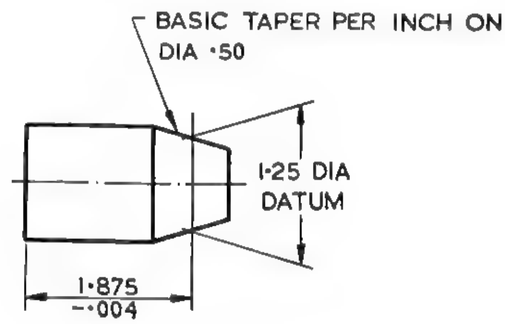


(*a*) Drawing specification.

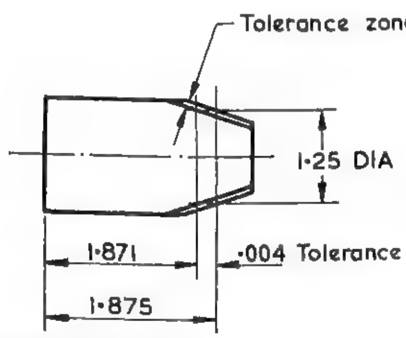


(*b*) Tolerance diagram for (*a*). Variations in the rate of taper are limited by the 0·002 diametral tolerance.

Fig. 86. Tolerancing a tapered object by the basic taper (or angle) method, using a datum distance.



(*a*) Drawing specification.



(*b*) Tolerance diagram for (*a*). Variations in the rate of taper are limited by the 0·004 tolerance on distance.

Fig. 87. Tolerancing a tapered object by the basic taper (or angle) method, using a datum diameter.

CLAUSE 18 (*cont.*)

The basic taper (or angle) method using a datum distance as illustrated in Fig. 86 (*a*) is particularly suitable for slow tapers (Fig. 88) and for the dimensioning of cones which need clearance on assembly (Fig. 89).

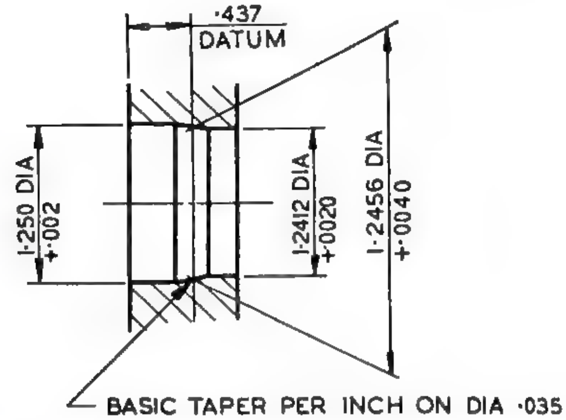
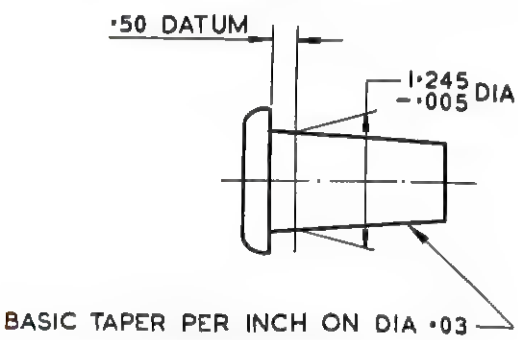
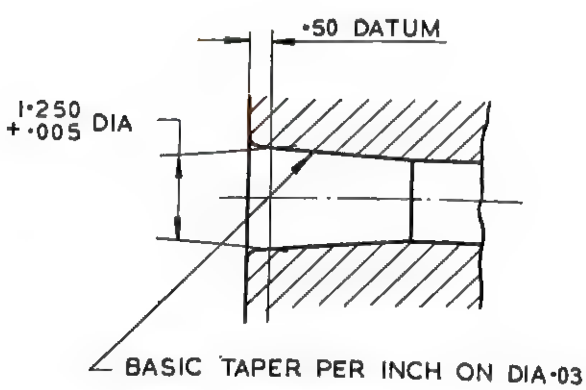


Fig. 88. Basic taper method of tolerancing gradual or slow internal tapers



(*a*) Cartridge case.



(*b*) Chamber.

Fig. 89. Basic taper (or angle) method of tolerancing components where tapered features need clearance on assembly

Fast tapers and tapers which determine endwise location should be dimensioned using datum diameters as shown in Figs. 90 and 91.

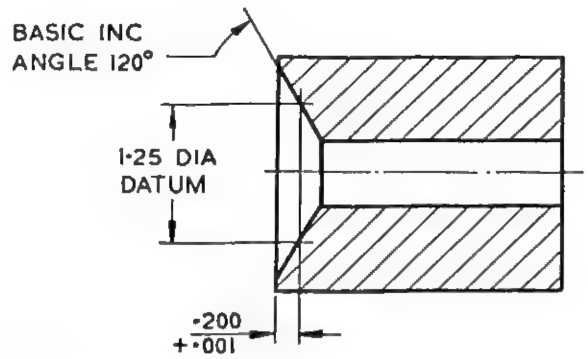
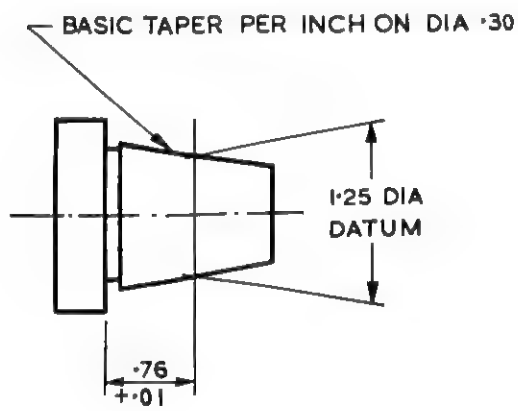
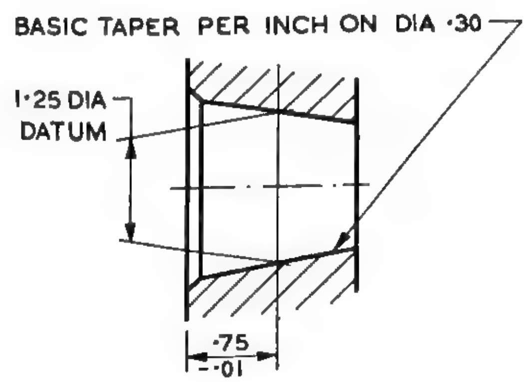


Fig. 90. Basic taper (or angle) method of tolerancing a fast taper



(*a*)

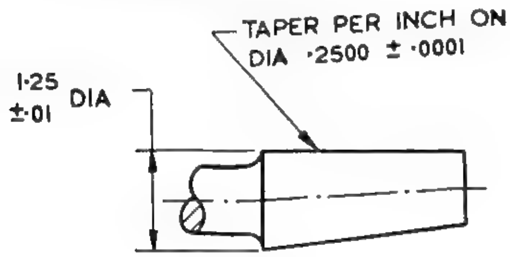


(*b*)

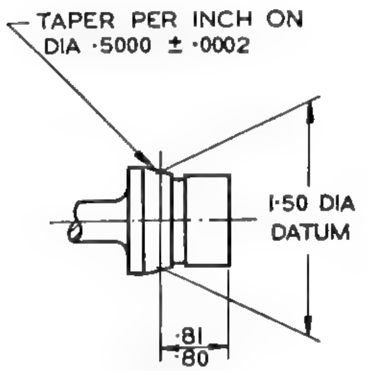
Fig. 91. Basic taper (or angle) method of tolerancing mating tapers which determine endwise location

CLAUSE 18 (*cont.*)

d. The toleranced taper (or angle) method. In this method a tolerance is applied directly to the rate of taper (or the included angle) independently of the tolerance which is specified for the size of the feature. Therefore, the tolerance of size applies only at the plane at which the dimension is shown on the drawing and *not* at every cross-sectional plane as is the case with the basic taper method. This method is used where the variation of taper (or angle) permitted by the basic taper (or angle) method would be too wide or, conversely, too restrictive.

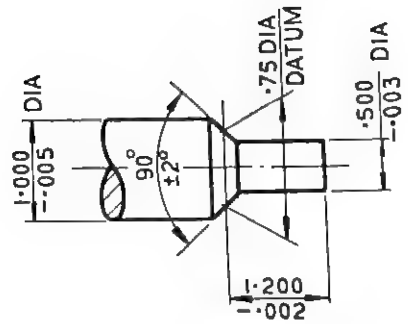


(*a*)

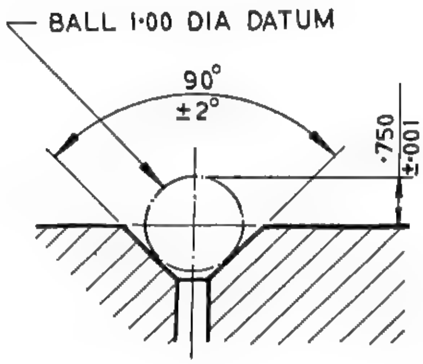


(*b*)

Fig. 92. Tolerancing tapered objects where the rate of taper is more important than size or location



(*a*)



(*b*)

Fig. 94. Alternative methods of tolerancing a tapered object where the location of a datum diameter is more important than the taper

e. Fitting to gauge or mating part. Where it is necessary to specify that a tapered surface must fit a gauge, or another component, notes such as those shown in Figs. 94 and 95 may be used.

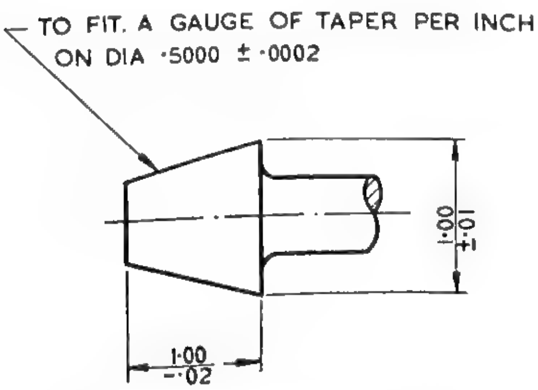


Fig. 93. Defining a taper by reference to a gauge

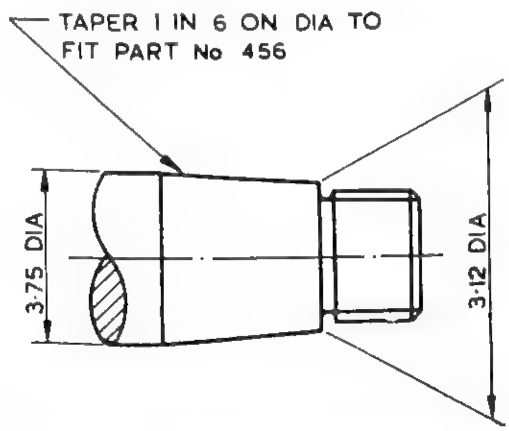


Fig. 95. Defining a taper by reference to its mating component

19. GEOMETRICAL TOLERANCES

Straightness, flatness, parallelism, squareness, angularity, concentricity, symmetry, position

a. Definition. A geometrical tolerance is the maximum permissible overall variation of form or position about that shown on the drawing. In other words, it is the width or diameter of a tolerance zone within which the surface, or the middle plane or axis of the feature, is to lie. It represents the *full* indicator movement in cases where testing with an indicator is applicable.

b. Application and interpretation. (i) Geometrical tolerances should be specified, where appropriate, for all requirements critical to functioning and interchangeability wherever it is doubtful that ordinary or established workshop technique and equipment can be relied upon to achieve the required standard of accuracy. Tables 2 to 7 show, in the first column, how to specify geometrical tolerances on drawings by means of standard abbreviated notes. The tolerance diagrams in the second column show how these geometrical tolerances are interpreted.

In the case of positional tolerances (see Table 7), it will be seen that the dimensions which define the true positions of the features are marked 'TP' (i.e., true position). This is to distinguish them from other dimensions which may be toleranced by a general tolerance note. Where a general tolerance note is not used and all dimensions other than auxiliary and datum dimensions and those defining true position are individually toleranced, these distinguishing letters (TP) may be omitted. In such cases a general note may be used to indicate that all untoleranced dimensions define true positions (TP). The letters 'TP' are also used when dimensioning profiles subject to profile tolerances (see Clause 21*b* (i), page 76).

In each of the examples shown in Tables 2 to 7, the geometrical tolerance is specified for the entire surface, axis or median plane regardless of its area or length. However, there are instances where it is preferable to specify it per unit area, or length (e.g. STR TOL 0·02 dia. per 72 inch length; SQ TOL 0·004 wide per foot run). Particular examples are the straightness of long tubes, and the flatness, squareness or parallelism of extensive functional surfaces in machine tools.

It must be clearly understood that when geometrical tolerances are expressed as shown in Tables 2 to 7, **they are to be observed regardless of the actual finished sizes of the features concerned.** However, there are many applications where the amount of geometrical tolerance can vary according to the finished size of the features. The method of specifying geometrical tolerances for such applications is described in sub-clause *c.* (page 65).

(ii) The study of these tables will show that in spite of the various terms (straightness, concentricity, etc.) used to describe geometrical tolerances, it often happens that one type of geometrical tolerance will automatically limit other types of geometrical error. For example, the parallelism tolerance quoted in the example 'Parallelism 1' in Table 3, in requiring the surface to be contained within a tolerance zone bounded by two planes 0·003 in. apart, limits errors of flatness as well as those of parallelism. The same can be said of the note quoted in 'Squareness 1' in Table 3. It can also be shown that tolerances for symmetry automatically limit errors of flatness and parallelism and that concentricity tolerances limit errors of straightness of the axis of the features concerned.

(iii) It would be quite correct to use the word 'position' to describe geometrical tolerances for symmetry and concentricity, particularly where errors of squareness or angularity must be accommodated within the one tolerance zone. Figure 107 shows a particular example in which the words 'concentricity and squareness' could be replaced by the one word 'position'; and it is anticipated that this word may eventually be used in all such cases. The word 'position' should always be interpreted and used in its widest sense, that is to say, by regarding it as involving not only locational relationships, but also angular relationships. In the example in 'Position 3' in Table 7, one of the datum features is the face B and it is, therefore, a requirement that the tolerance cylinders for the positions of the two holes must be perpendicular to this face as well as in their true position, relative to each other and hole A. Thus the positional tolerance in this example will automatically limit errors of squareness of the holes with face B and errors of parallelism with each other. A further example which illustrates the embracing nature of positional tolerances is shown in Fig. 96.

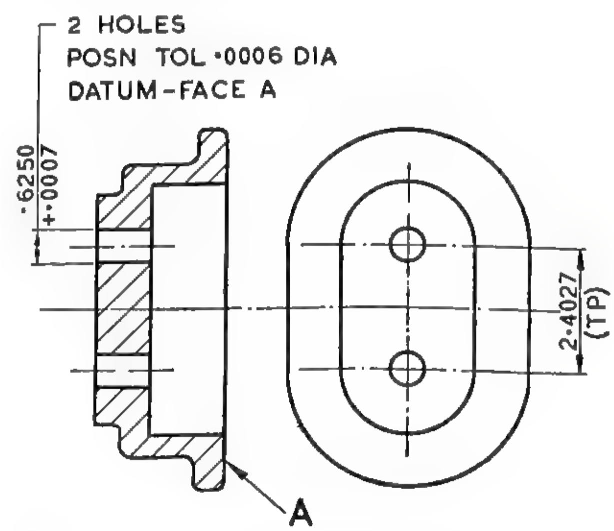
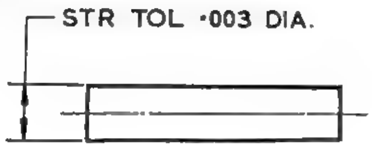
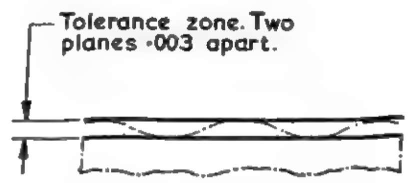
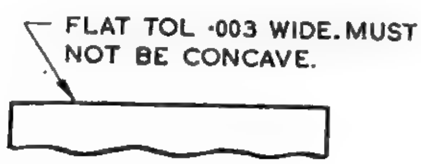
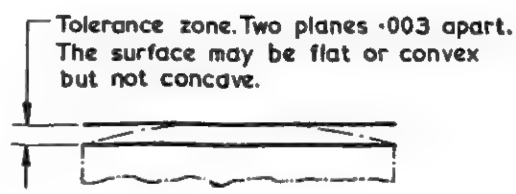


Fig. 96. Positional tolerance controlling squareness and parallelism in addition to centre distance

CLAUSE 19 (*cont.*)

TABLE 2. TOLERANCES FOR STRAIGHTNESS AND FLATNESS

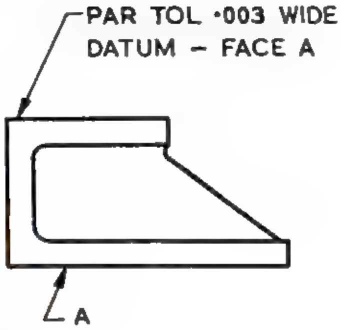
EXAMPLES	INTERPRETATIONS NOTE. The surface, or axis of the feature must lie within the tolerance zones shown in these diagrams.
STRAIGHTNESS 	
FLATNESS 1 	
FLATNESS 2 	

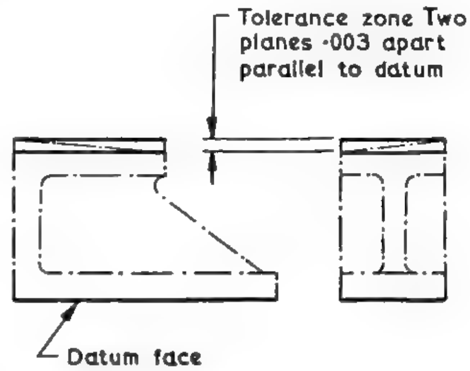
CLAUSE 19 (*cont.*)

TABLE 3. TOLERANCES FOR PARALLELISM

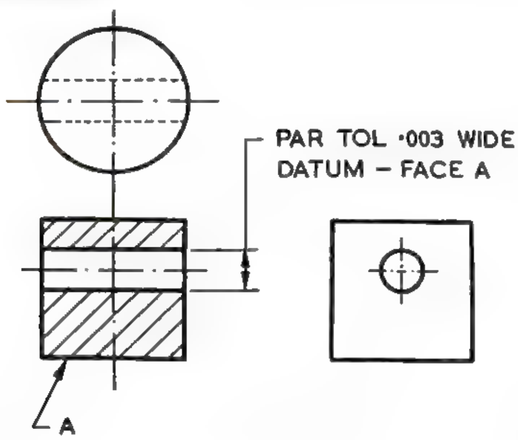
EXAMPLES	INTERPRETATIONS NOTE. The surface, or axis of the feature must lie within the tolerance zones shown in these diagrams.

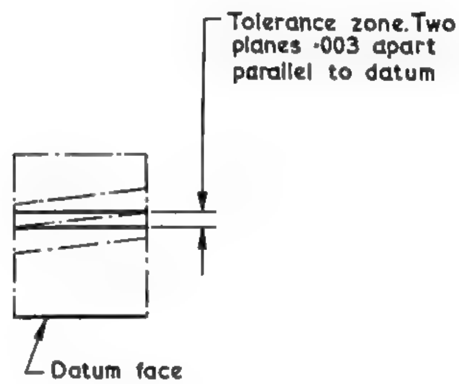
PARALLELISM 1



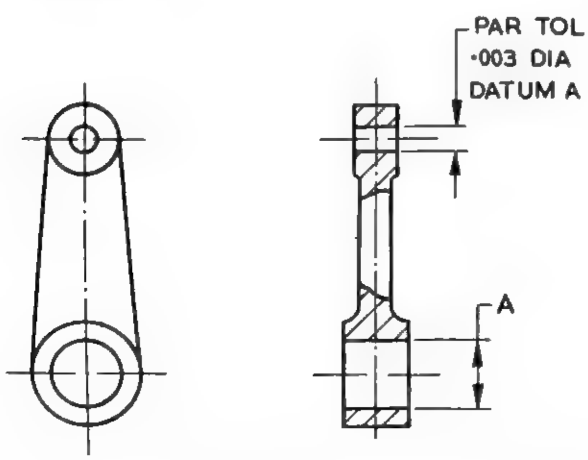


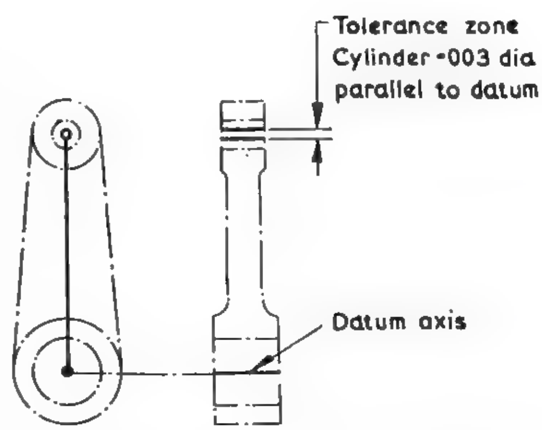
PARALLELISM 2





PARALLELISM 3



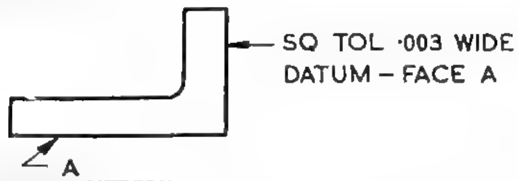


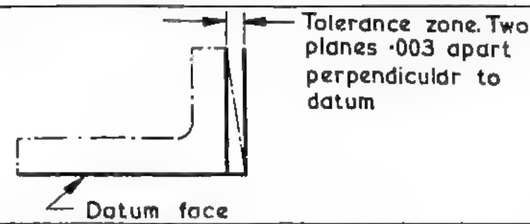
CLAUSE 19 (*cont.*)

TABLE 4. TOLERANCES FOR SQUARENESS AND ANGULARITY

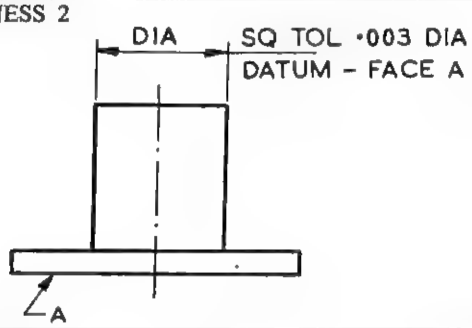
EXAMPLES	INTERPRETATIONS NOTE. The surface, or axis of the feature must lie within the tolerance zones shown in these diagrams.

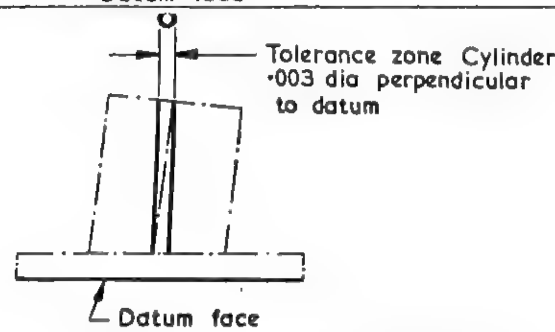
SQUARENESS 1



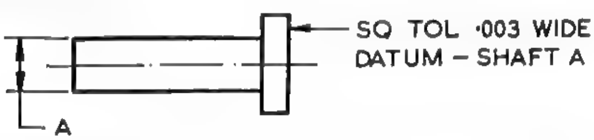


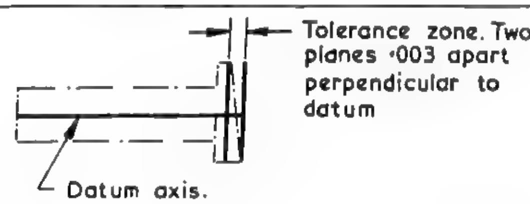
SQUARENESS 2



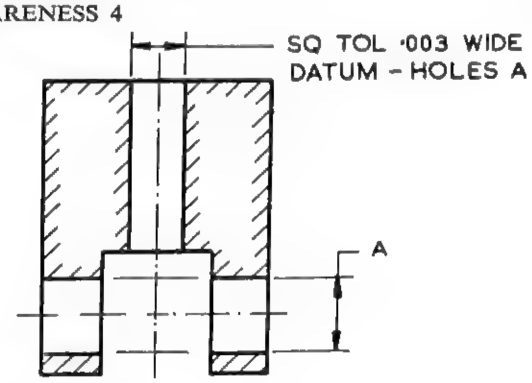


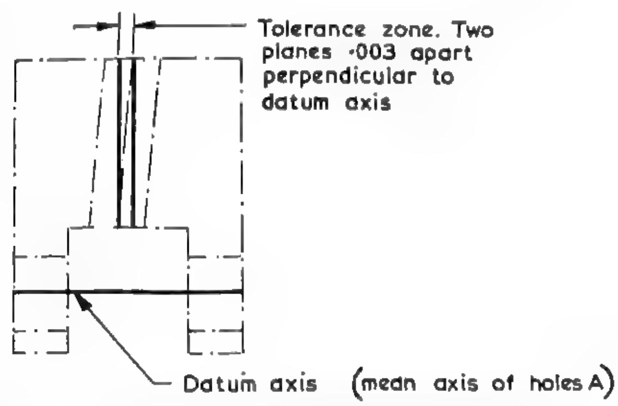
SQUARENESS 3



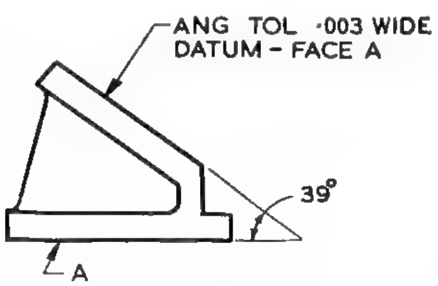


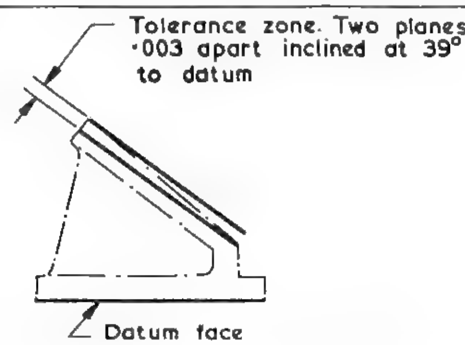
SQUARENESS 4





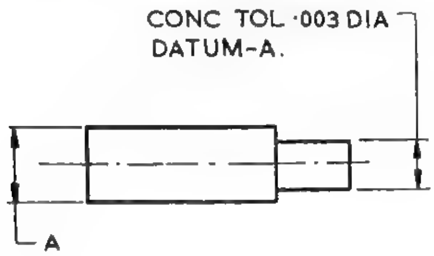
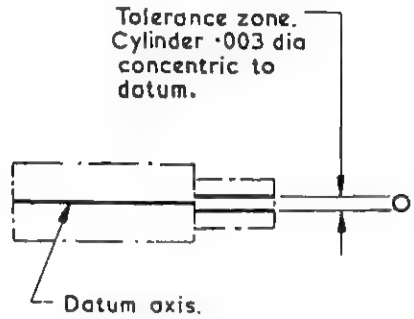
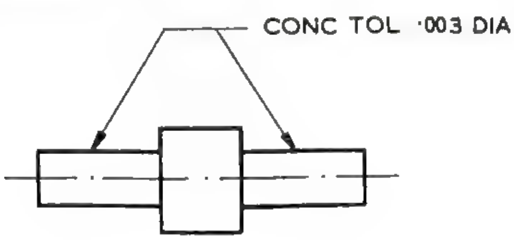
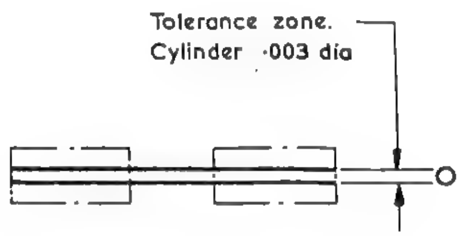
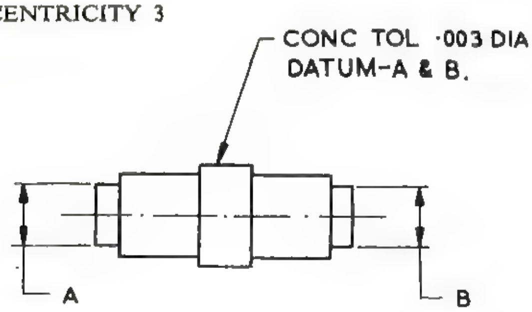
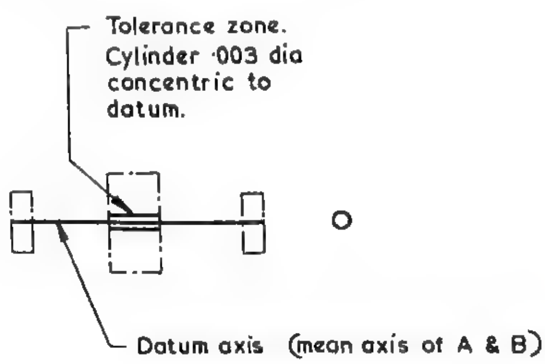
ANGULARITY





CLAUSE 19 (*cont.*)

TABLE 5. TOLERANCES FOR CONCENTRICITY

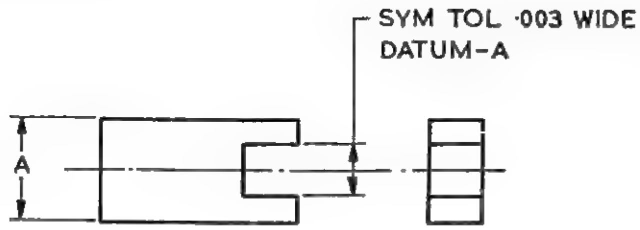
<table>
<tr><th>EXAMPLES</th><th>INTERPRETATIONS
NOTE. The axis of the feature must lie within the tolerance zone shown in these diagrams.</th></tr>
<tr><td>CONCENTRICITY 1

</td><td>
</td></tr>
<tr><td>CONCENTRICITY 2

</td><td>
</td></tr>
<tr><td>CONCENTRICITY 3

</td><td>
</td></tr>
</table>

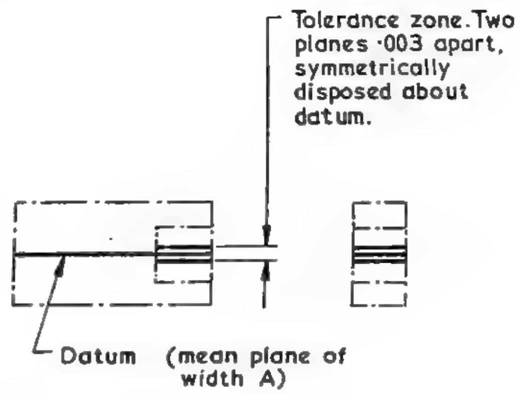
CLAUSE 19 (*cont.*)

TABLE 6. TOLERANCES FOR SYMMETRY

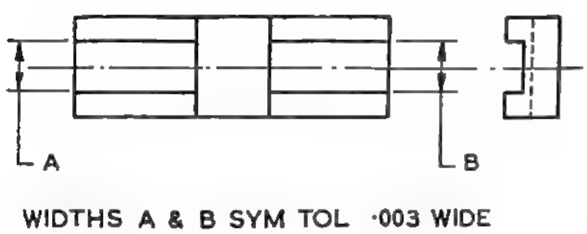
EXAMPLES	INTERPRETATIONS
	NOTE. The axis or middle plane of the feature must lie within the tolerance zones shown in these diagrams.

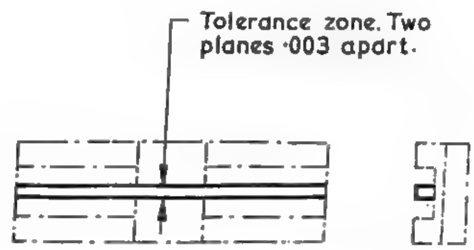
SYMMETRY 1



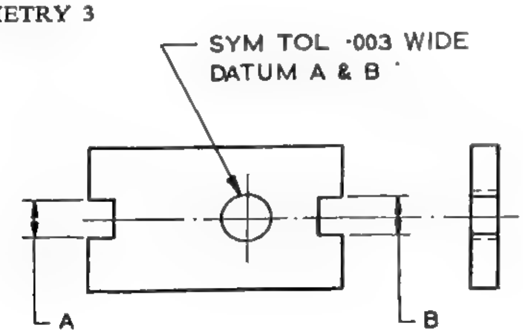


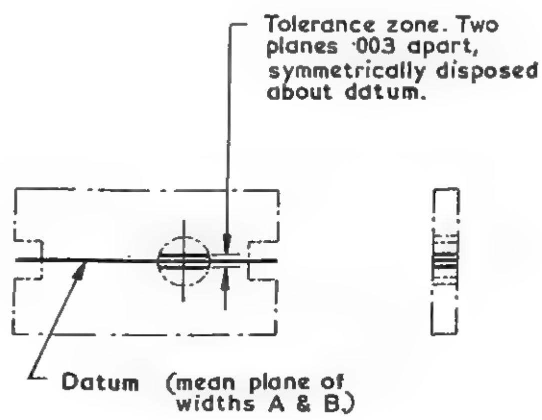
SYMMETRY 2





SYMMETRY 3



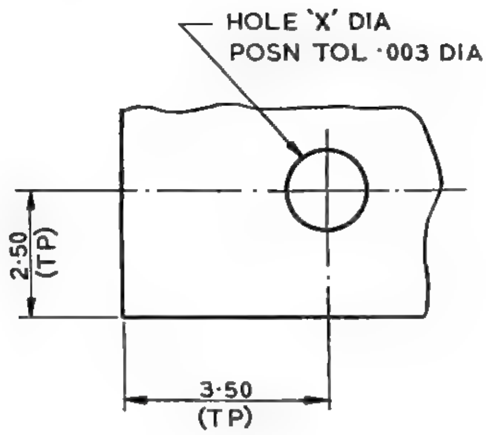


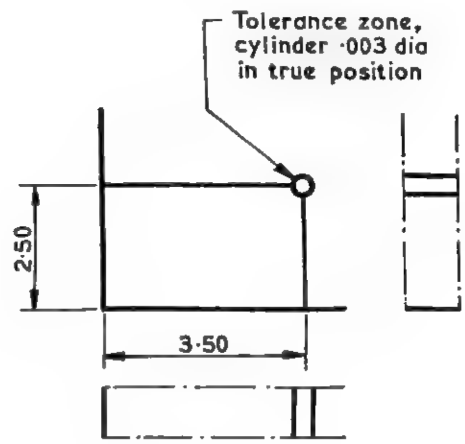
CLAUSE 19 (*cont.*)

TABLE 7. TOLERANCES FOR POSITION

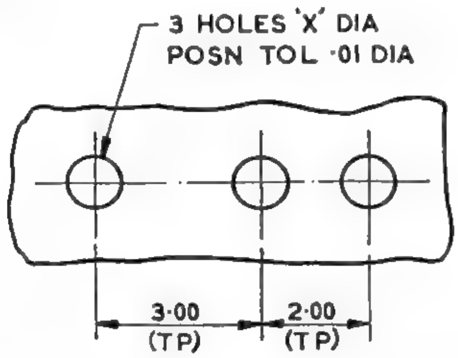
EXAMPLES	INTERPRETATIONS
	Note. The axis of each feature must lie within the tolerance zones shown in these diagrams.

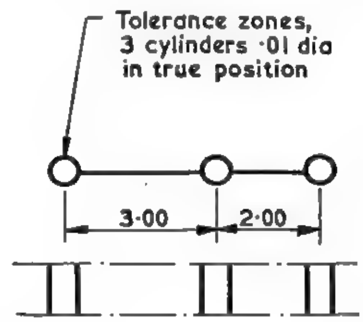
POSITION 1



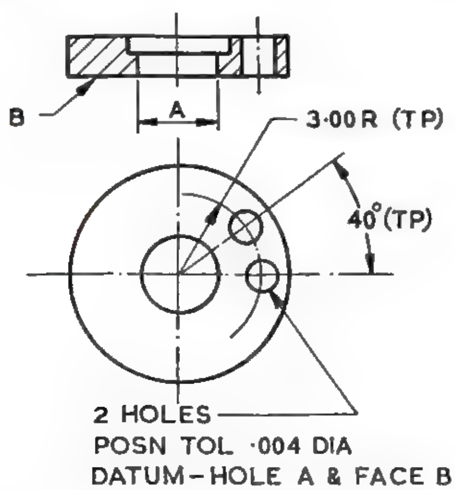


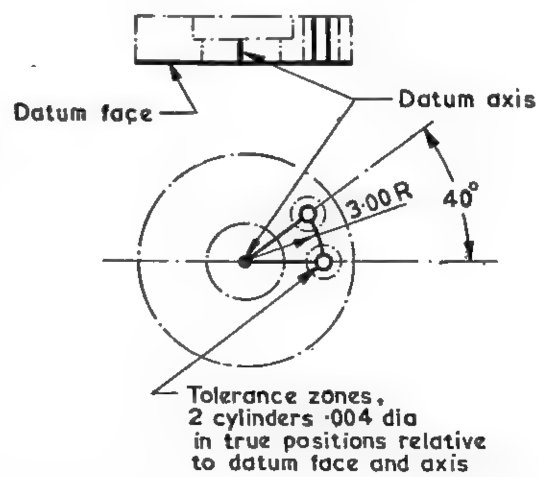
POSITION 2



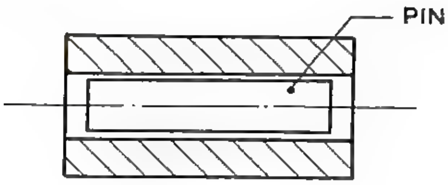


POSITION 3

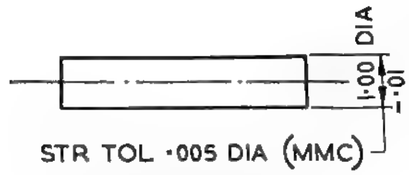




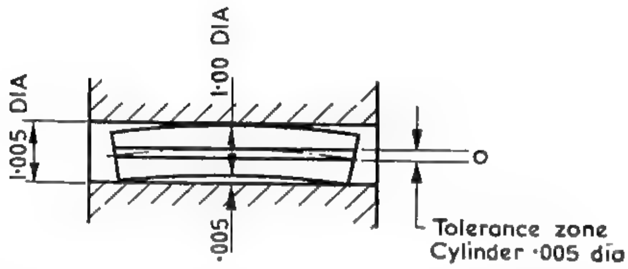
CLAUSE 19 (*cont.*)



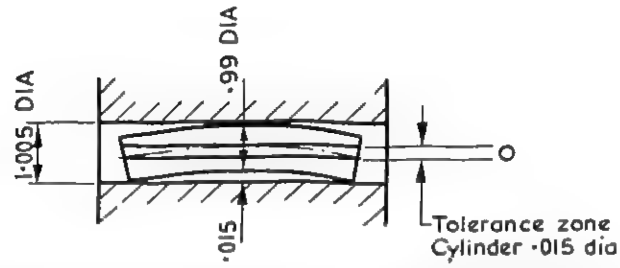
(*a*) Assembly.



(*b*) Detail drawing of pin.

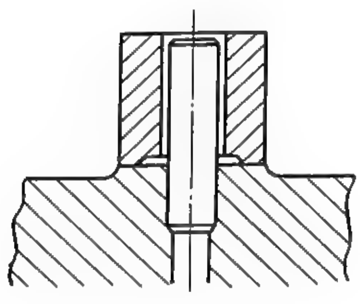


(*c*) Maximum metal pin shown in gauge. The effective straightness tolerance is limited to the stated tolerance of 0·005 dia.

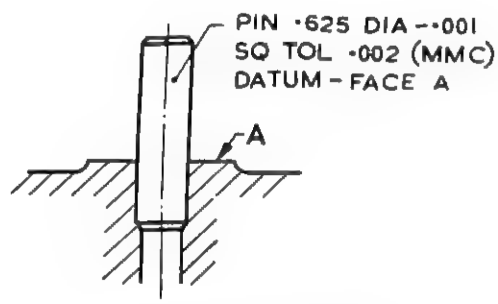


(*d*) Minimum metal pin shown in gauge. The effective straightness tolerance is limited to the stated tolerance of 0·015 dia.

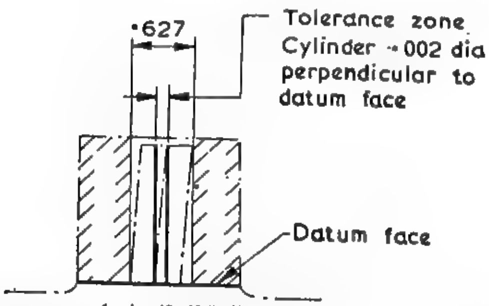
Fig. 97. Straightness tolerance specified with reference to maximum metal condition



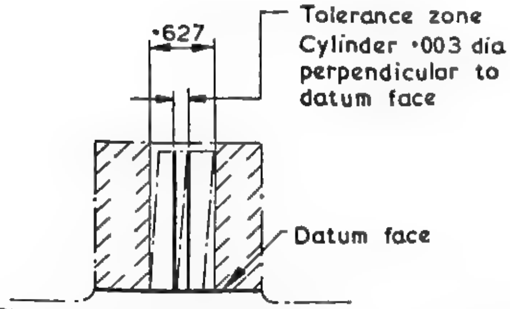
(*a*) Assembly.



(*b*) Detail drawing.



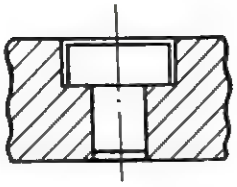
(*c*) Maximum metal pin (0·625 dia.) shown in gauge. Effective squareness tolerance 0·002 dia.



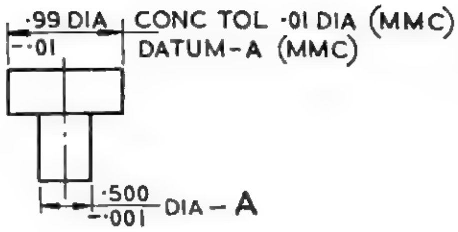
(*d*) Minimum metal pin (0·624 dia.) shown in gauge. Effective squareness tolerance 0·003 dia.

Fig. 98. Squareness tolerance specified with reference to maximum metal condition

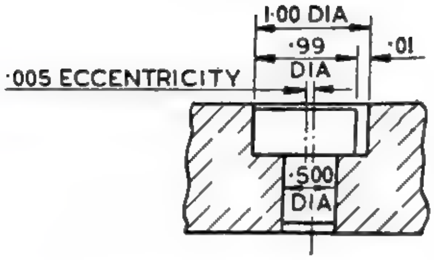
CLAUSE 19 (*cont.*)



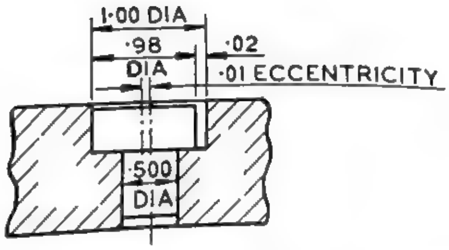
(*a*) Assembly.



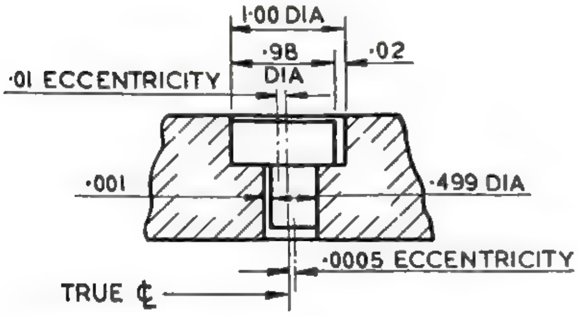
(*b*) Detail drawing.



(*c*) Pin with head and shank on maximum metal limits shown in gauge. The effective concentricity tolerance is limited to stated tolerance of 0·01 dia.



(*d*) Pin with head on minimum metal limit, and shank on maximum metal limit, shown in gauge. Effective concentricity tolerance = stated conc. tol. (0·01) + tolerance on size of head (0·01) = 0·02 dia.

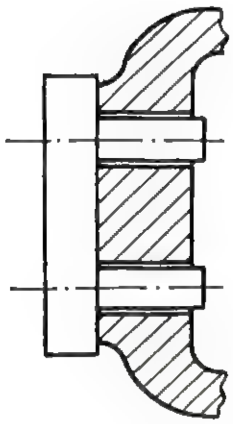


(*e*) Pin with head and shank on minimum metal limits shown in gauge. Effective concentricity tolerance = stated conc. tol. (0·01) + tolerance on size of head (0·01) + tolerance on size of shank (0·001) = 0·021 dia.

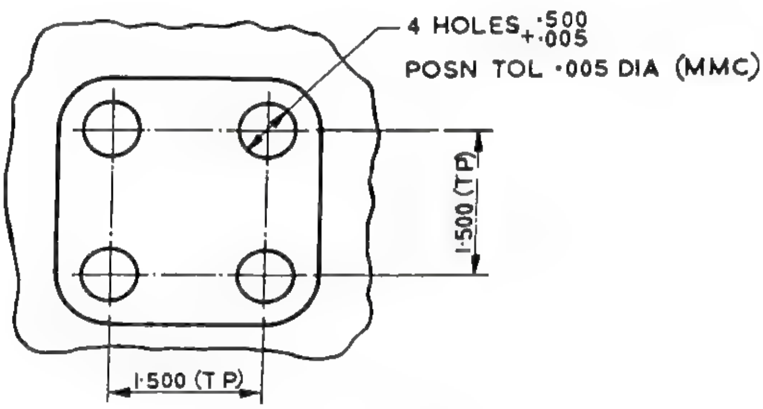
Fig. 99. Concentricity tolerance specified with reference to maximum metal condition

CLAUSE 19 (*cont.*)

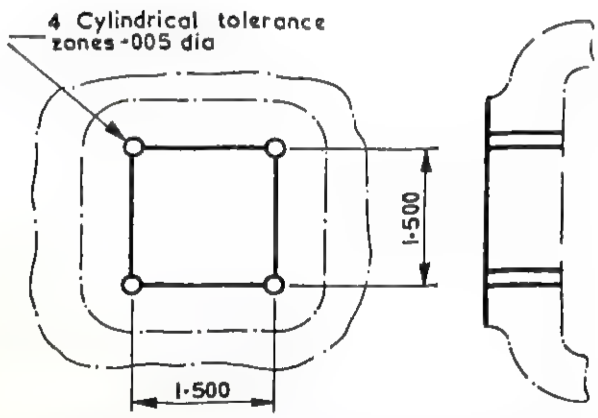
c. (i) *Relationship between actual size and geometrical accuracy*. The possibility of the free assembly of two components as shown in Figs. 97 (*a*), 98 (*a*), 99 (*a*) or 100 (*a*) is dependent on the combined effect of the actual sizes and the errors of straightness, squareness, concentricity, position, etc. When a feature has errors of form or position its size is virtually altered; the size of a hole is virtually reduced and the size of a shaft virtually increased. The worst condition for assembly occurs when the mating features are on their maximum metal limits of size, and the maximum errors permitted by the geometrical tolerances are present. Thus, in the worst condition, the pin shown in Fig. 97 (*b*) will just be able to enter a truly cylindrical hole 1·005 diameter (Fig. 97 (*c*)). However, if the pin is finished to a size other than its maximum metal limit, say for example 0·99 diameter, there will be an increase in the clearance between it and its mating feature, and a larger straightness tolerance than that stated on the drawing could be allowed without endangering free assembly (Fig. 97 (*d*)). This increase of geometrical tolerance is equal to the difference between the specified maximum metal limit of size and the actual finished size of the feature. Figs. 98 and 99 show further examples of requirements for squareness, concentricity and position, for which increased geometrical tolerances are permissible when the actual sizes of the features are away from their maximum metal limits of size. Wherever this increase in geometrical tolerance can be allowed, the letters MMC (i.e., maximum metal condition) should be included in the tolerance notes as in Figs. 97 (*b*), 98 (*b*), 99 (*b*) and 100 (*b*).



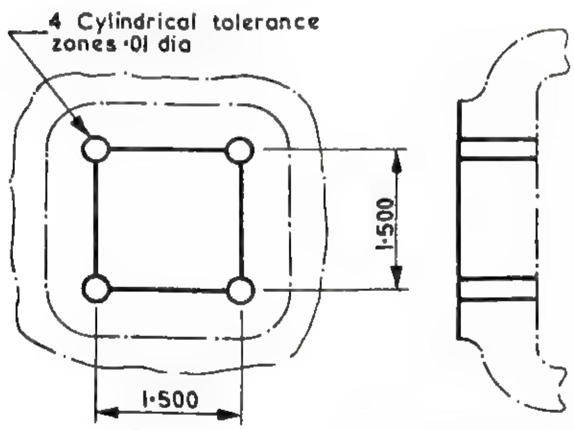
(*a*) Assembly.



(*b*) Detail drawing.



(*c*) Tolerance diagram for (*b*) where holes are in their maximum metal condition (0·500 dia.).



(*d*) Tolerance diagram for (*b*) when holes are in their minimum metal condition (0·505 dia.).

Fig. 100. Positional tolerance specified with reference to maximum metal condition

CLAUSE 19 (*cont.*)

(ii) Where it is necessary to state that any errors of straightness, concentricity, squareness, etc., are to be contained within the maximum metal limits of the features, the form of note shown in Figs. 101, 102 and 103 should be used. Such notes would mean that, if the features are finished everywhere on their maximum metal limits of size, they must be perfect in form (i.e., truly straight, square, concentric, etc.), but errors of form are permissible if the features are away from their maximum metal limits of size in the direction of minimum metal.

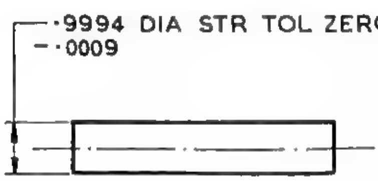


Fig. 101. Tolerance for straightness included within limits of size to ensure assembly

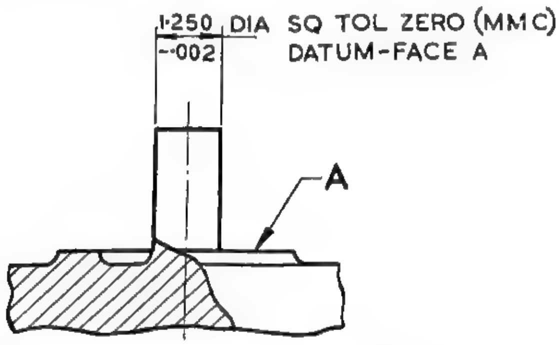


Fig. 102. Tolerance of squareness included in limits of size

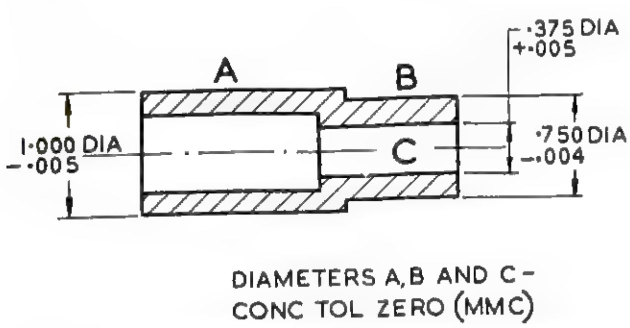


Fig. 103. Tolerances for concentricity included in limits of size

NOTE 1. For practical reasons the above method of specifying geometrical tolerances should not be used for positional requirements.

NOTE 2. When using these methods the tolerance specified for the size of the feature must not be less than the sum of the minimum practical tolerances for size and form.

NOTE 3. Zero geometrical tolerances cannot be specified except in relation to the maximum metal condition. To do otherwise would be to demand perfection.

d. Use of general notes for geometrical tolerances. Where appropriate, geometrical tolerances may be specified by general notes as in the typical examples given below.

'Except where otherwise stated, machined surfaces are to be square with each other within a tolerance 0·003 wide.'

'Except where otherwise stated, machined surfaces are to be parallel to face A within a tolerance 0·005 wide.'

'Except where otherwise stated, machined surfaces are to be square with each other within a tolerance 0·004 wide per foot run.'

'Except where otherwise stated, each machined diameter is to be concentric with feature A within a tolerance of 0·005 dia.'

e. Advantages of positional tolerances over toleranced centre distances. Under certain conditions the use of positional tolerances offers some advantage over toleranced centre distances, particularly for the positioning of features on pitch circles or otherwise located by angular dimensions. The main advantages of the use of positional tolerances are :

(i) It corresponds to the control exercised by position gauges and to the distribution of errors which normally arise in production.

(ii) It simplifies the tolerancing of complex parts.

(iii) It permits the use of chain dimensioning without involving the accumulation of tolerances.

(iv) It facilitates the specification of different tolerances for the positions of each of a number of features lying on a common centre line or located by the same dimension.

(v) It facilitates the assessment of clearances between the features of mating components because equal deviations are permitted in all directions.

On the other hand, toleranced centre distances are usually just as satisfactory where only two features need to be related ; they also permit the amount of tolerance to be different in two directions (Fig. 60).

f. Some typical applications. The following text and figures illustrate suitable methods of expressing geometrical tolerances for some typical applications. It will be seen that there are two general methods of placing the requirements on the drawing, one which places the note adjacent to the feature concerned (Tables 2 to 7), and the other which uses a letter to identify the feature and then states the tolerance by means of a note or table in a convenient position on the drawing. When applying geometrical tolerances, the datum features (if any) should be indicated in the tolerance note or table except where the method of dimensioning makes them obvious as in the first examples in Table 7.

NOTE. The selection of datum features and the grouping of related features should be based primarily on the functional requirements of the design.

Fig. 104. This figure shows the application of concentricity tolerances to a rotary spindle which is located in its housing by means of rolling bearings mounted on the

CLAUSE 19 (*cont.*)

seatings C and D. These two seatings are used as the datum features for tolerancing the concentricity of the other portions of the spindle (G, H and K) which carry gear wheels and other components.

Fig. 105. An example of a stated concentricity tolerance specified in relation to the maximum metal condition of the features is shown in Fig. 105. In this example the concentricity of the two holes is toleranced to ensure interchangeable assembly with the pin shown in Fig. 101.

It should be noted that, in this example, neither of the holes can be regarded as a datum for the group.

The interpretation in Fig. 105 (*c*) shows that the holes are required to be concentric with their mean axis to within 0·0006 dia. if they are both in their maximum metal condition. If, however, they are not in their maximum metal condition this concentricity tolerance is increased as explained in Sub-clause *c* (i), page 65.

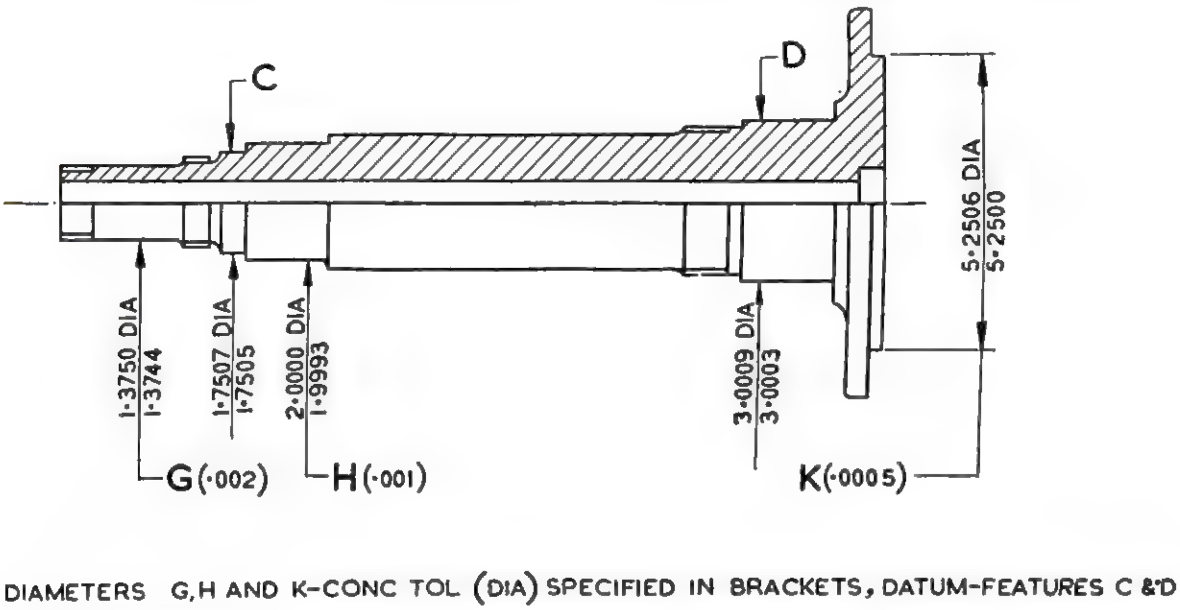
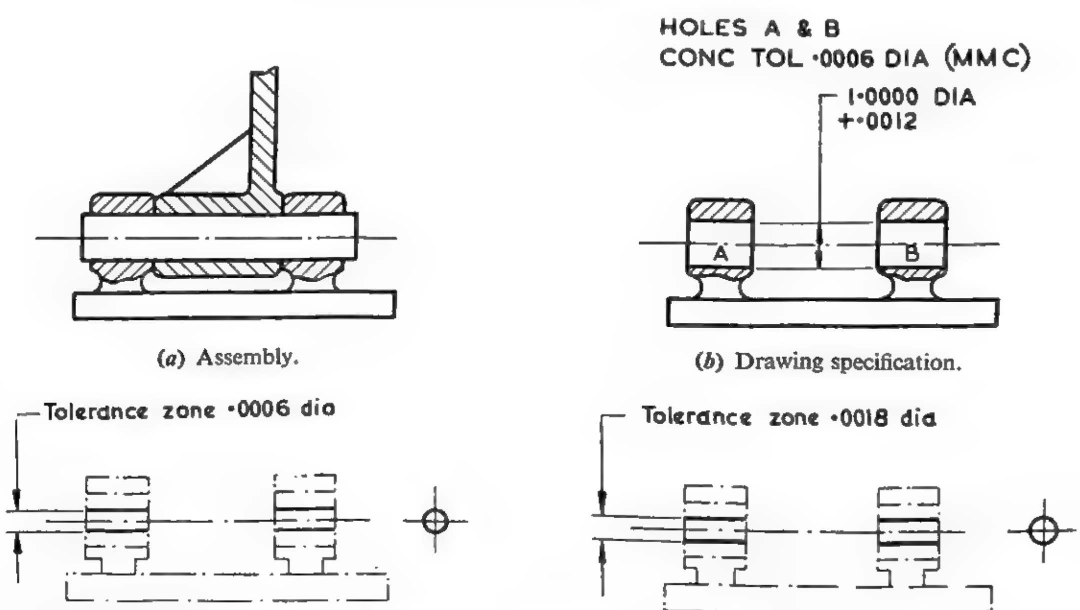


Fig. 104. Tolerance notes designating features by letters



(*a*) Assembly.

(*b*) Drawing specification.

(*c*) Interpretation of requirement of concentricity in maximum and minimum metal conditions. The axes of holes A and B must lie within the tolerance zones.

Fig. 105. Tolerancing concentricity of holes to ensure assembly with pin. For detail specification of pin see Fig. 101

CLAUSE 19 (*cont.*)

Fig. 106. It is often necessary for features to be toleranced for angularity, squareness or parallelism with relation to more than one datum feature. Fig. 106 shows a typical example in which the squareness of a face is toleranced in relation to two datum faces.

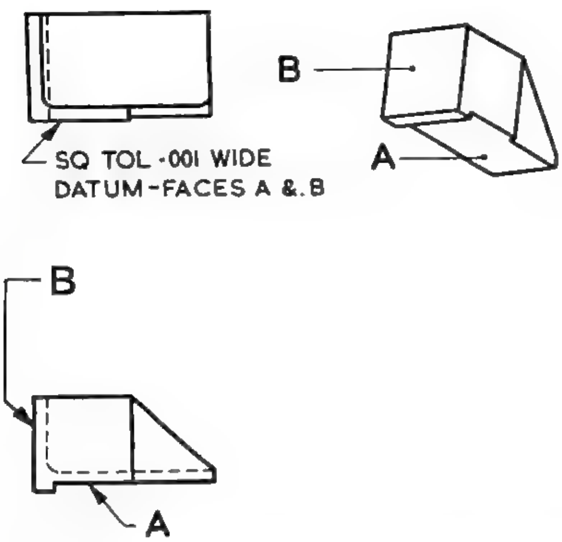


Fig. 106. Face toleranced for squareness to two datum faces

Fig. 107. Tolerances of squareness are often conveniently combined with other geometrical tolerances. A typical example is shown in Fig. 107 which shows a shaft extension the journal of which must run true with the main shaft within certain limits of error. A combined tolerance of concentricity and squareness is therefore applied to the journal, using recess A and face B together as a datum system. It would be equally correct to refer to this tolerance as a positional tolerance.

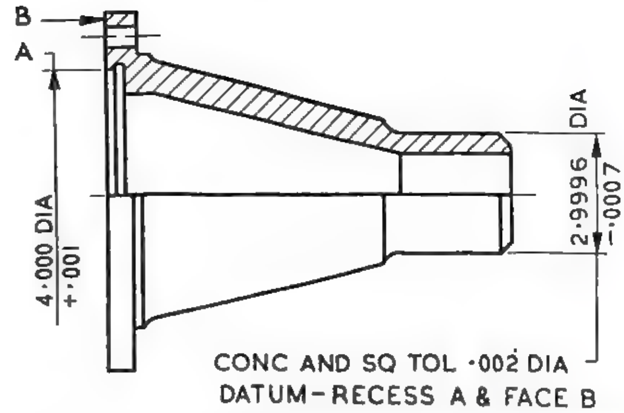


Fig. 107. Use of combined concentricity and squareness note

Fig. 108. A further example is shown in Fig. 108 in which a group of features must be truly concentric and square to a seating face, when in their maximum metal condition, in order to ensure satisfactory assembly and functioning.

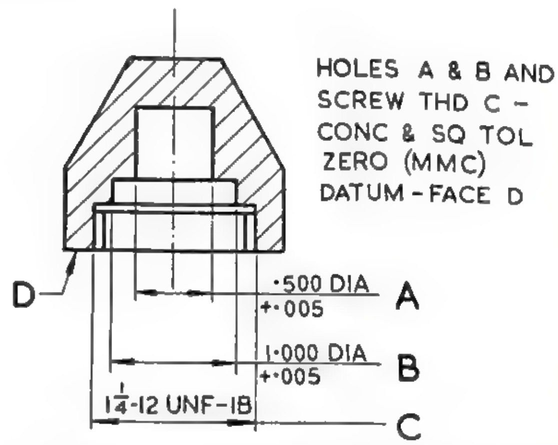


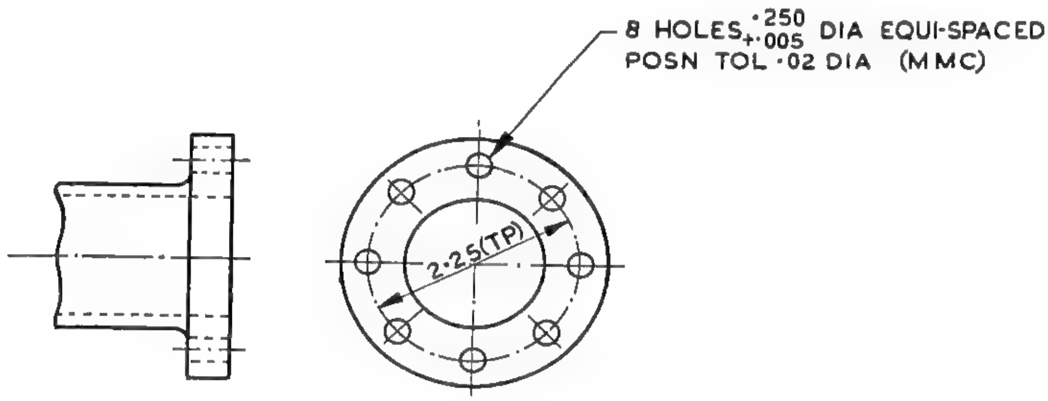
Fig. 108. Combined tolerance for concentricity and squareness included in limits of size

CLAUSE 19 (*cont.*)

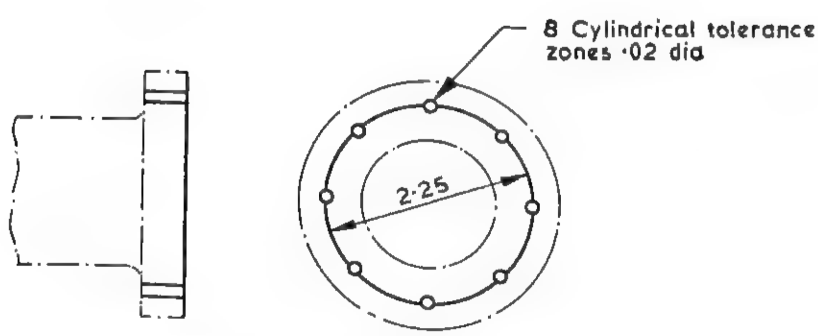
Fig. 109. Where all the features of a group can be allowed the same positional tolerance with relation to each other, a note such as that shown in Fig. 109 (*a*) may be used. Such a note limits the displacement of each of the features, relative to each other, in all directions. The interpretation shown in Fig. 109 (*b*) shows the positional tolerance zones for the maximum metal condition of the holes. When any one of the holes is finished away from its maximum metal limit of 0·250 dia., the diameter of its tolerance zone is automatically increased as described in Sub-clause (*c*) (i), page 65.

Fig. 110. A group of holes toleranced in relation to a datum spigot and datum face is shown in Fig. 110 (*a*). The tolerance diagram in Fig. 110 (*b*) shows the positional requirements for the maximum metal condition of the holes and datum spigot. If any hole is finished away from its maximum metal limit there will, in effect, be an increase of positional tolerance for that hole. Furthermore, if the datum spigot is finished away from its maximum metal limit of 6·00 diameter, there will be a further increase of positional tolerance for all the holes relative to the datum spigot but not in relation to each other.

Fig. 111. Positional tolerances may be applied to other than plain cylindrical features such as holes or pins. Fig. 111 shows an example in which two slots and a tongue are toleranced for position in relation to a datum bore and keyway by positional tolerances. The positional tolerance for the slots requires the median plane of each slot to lie between two planes 0·005 apart which are equi-disposed about the true positions. Similarly, the positional tolerance for the tongue requires the median plane of the tongue to lie inside a tolerance zone 0·01 wide equi-disposed about its true position. Here again, the positional tolerances apply to the features in their maximum metal condition and when any of the features, datum or otherwise, are finished away from their maximum metal limits an increase of positional tolerance is permitted.



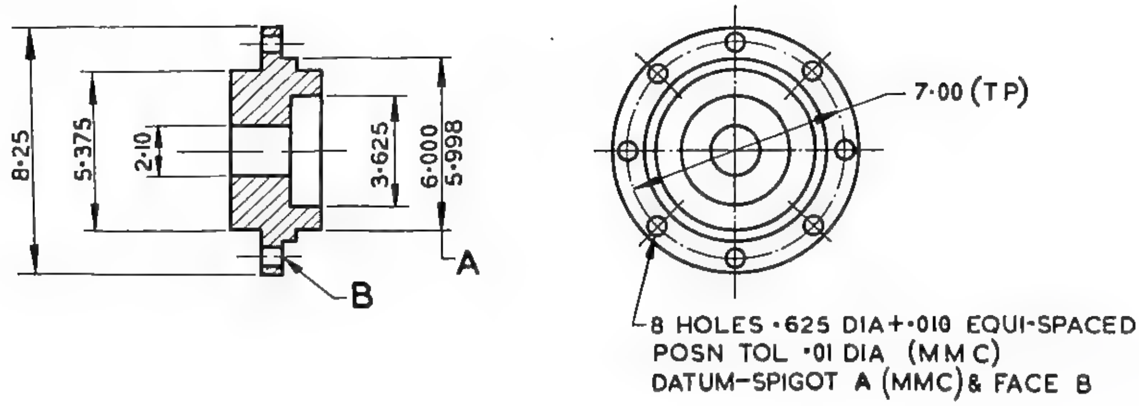
(*a*) Drawing specification.



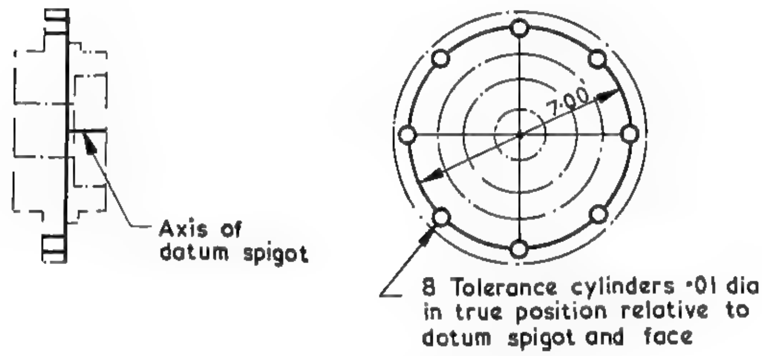
(*b*) Interpretation of positional tolerance in (*a*).

Fig. 109. Tolerancing the relative position of holes by positional tolerance note

CLAUSE 19 (*cont.*)



(*a*) Drawing specification.



(*b*) Interpretation of positional tolerance in (*a*).

Fig. 110. Tolerancing the relative position of holes with reference to a datum feature

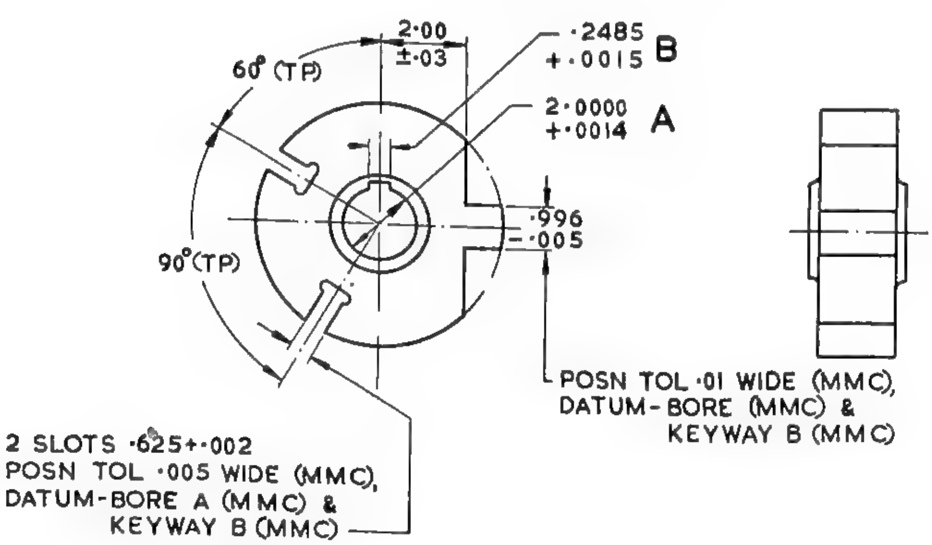
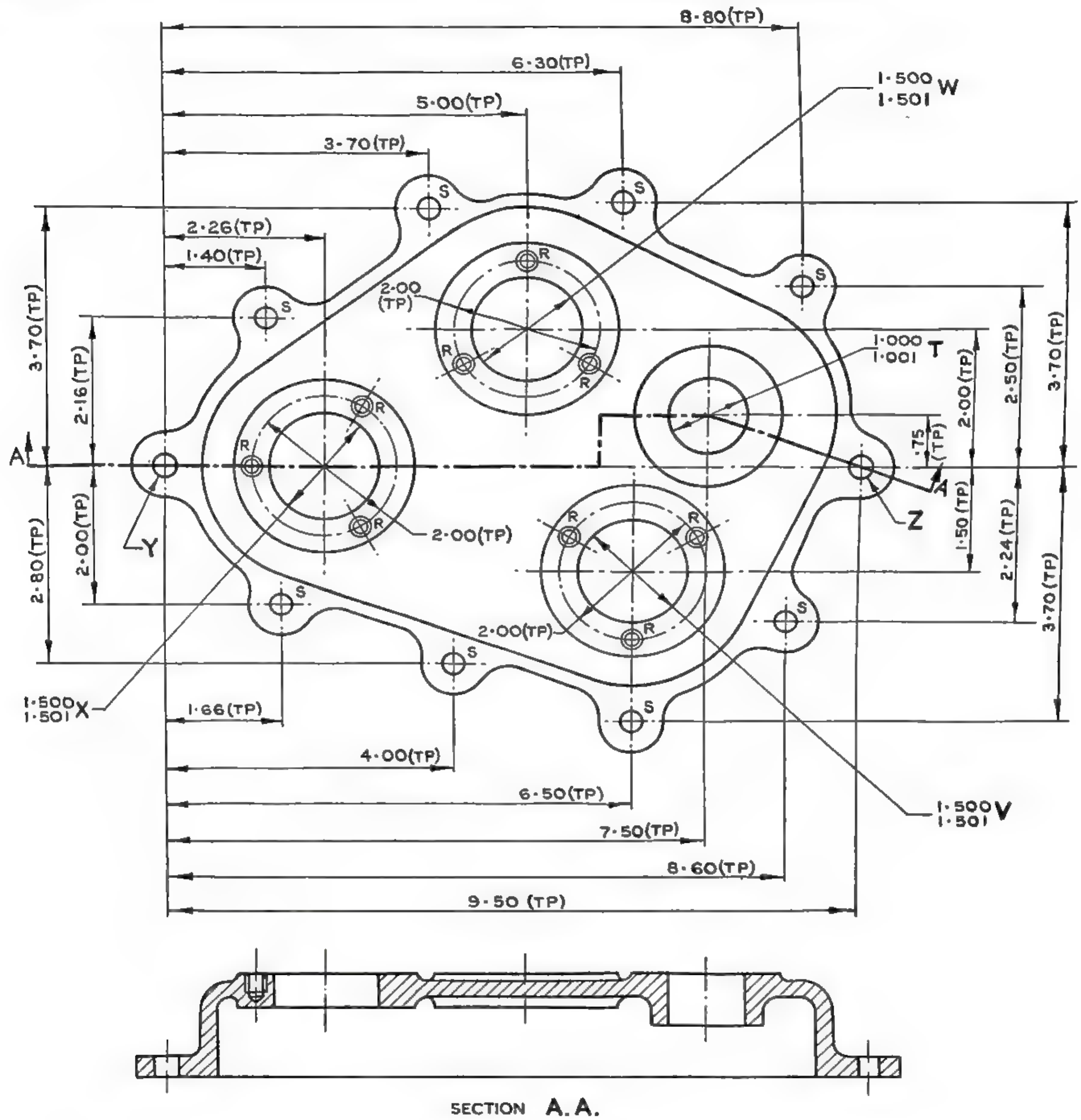


Fig. 111. Positional tolerances applied to rectangular features

CLAUSE 19 *(cont.)*

Fig. 112. Where a component contains a number of features which fall into groups, determined from consideration of the function of the component, positional tolerances can be used to relate each of the groups to each other as necessary, and also to tolerance the position of the features within each group independently of the features of other groups. This is illustrated in Fig. 112. In this example, the holes Y and Z are used to locate the part in the assembly and, therefore, these two holes act as datum features for the location of the securing bolt holes S and the holes marked T, V, W and X which carry components that need to be accurately located in the assembly. Finally, each of the holes V, W and X serves as a datum for its associated tapped holes.



HOLES Y & Z. $\frac{3125}{3133}$ DIA. POSN TOL ·001 DIA. (MMC)

HOLES T, V, W & X POSN TOL ·003 DIA (MMC), DATUM – HOLES Y & Z (MMC)

HOLES S, $\frac{17}{64}$ DIA. —— POSN TOL ·010 DIA. (MMC), DATUM – HOLES Y & Z (MMC)

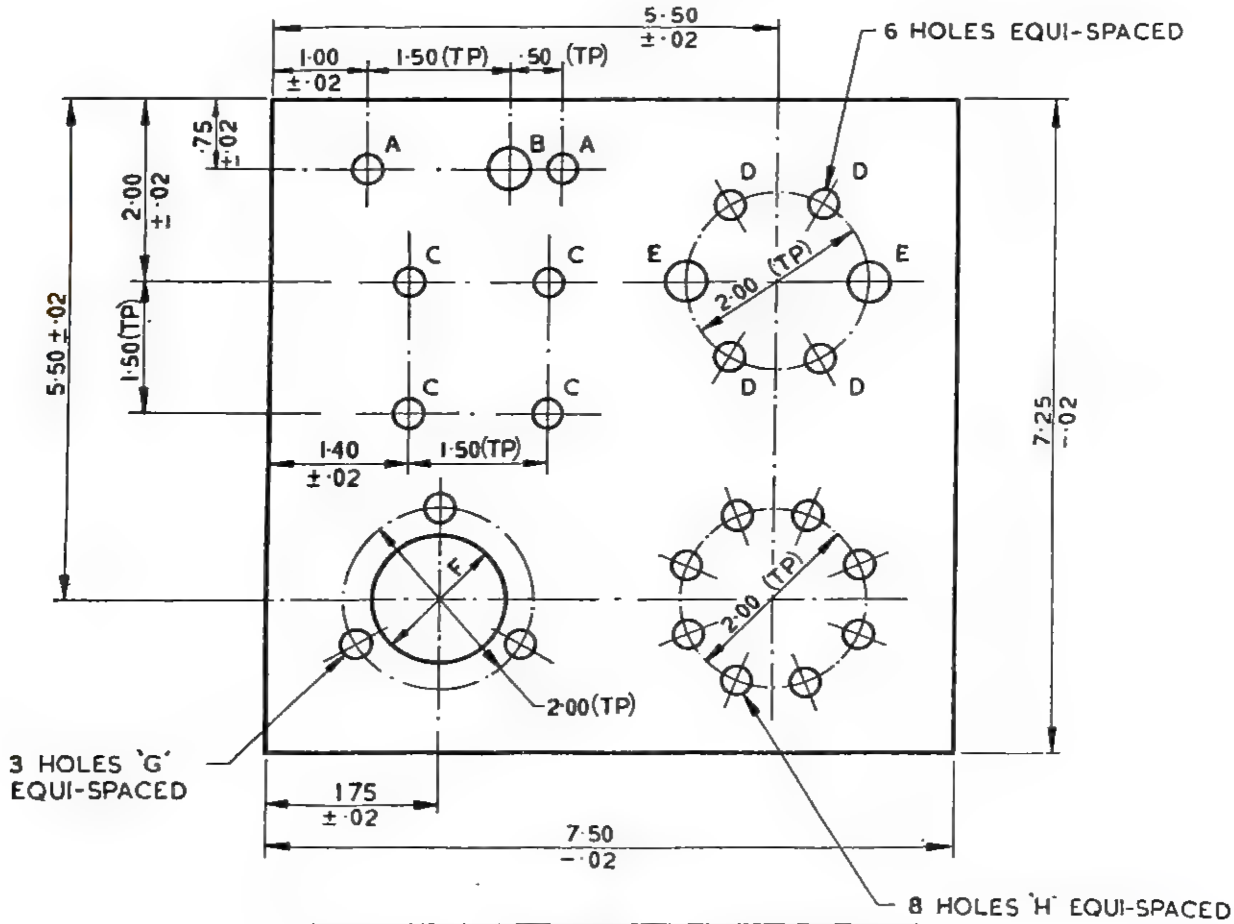
HOLES R, 3 GROUPS OF 3 HOLES, EQUI-SPACED, $\frac{1}{4}$ – 20 UNC – 2B, ·3 MIN LENGTH FULL THREAD POSN TOL ·010 DIA (MMC), DATUM – HOLE V, W OR X (MMC) (AS APPLICABLE)

Fig. 112. Positional tolerance notes using designating letters for features

CLAUSE 19 (*cont.*)

Fig. 113. Where the location of a group of features is relatively unimportant, while greater accuracy of position within the group is required, it is sometimes convenient to use both toleranced centre distances and positional tolerances. A typical example is that of the instrument panel shown in Fig. 113. In this example the positions of the holes within each group must be controlled so as to ensure the correct assembly of the components, but the positions of the components on the panel can be permitted to vary within wider limits.

Where such a combination of toleranced centre distances and positional tolerances is used, the toleranced centre distances are to be interpreted as referring only to the theoretical centre lines of each group. Each individual feature is then permitted to be displaced from its true position in its group within the positional tolerance for that feature.



GROUP	HOLES			POSN TOL (MMC)
	LETTER	DIA	No	
1	A	·250 +·005	2	01 DIA
	B	·50 +·01	1	
2	C	·250 +·005	4	·01 DIA
3	D	·250 +·005	4	·01 DIA
	E	·50 +·01	2	
4	F	1·50 +·01	1	DATUM
	G	·250 +·005	3	·01 DIA
5	H	·250 +·005	8	·01 DIA

Fig. 113. Relative positions of groups defined by toleranced centre distances given from sides of panel. Inter-relationships of holes in each group defined by 'TP' dimensions and positional tolerances. The positional tolerances are tabulated

CLAUSE 19 (*cont.*)

Fig. 114. It is advisable, in the interests of economy, to indicate where the angular location of a feature, or a group of features, is unimportant. In such cases the form of note shown in Fig. 114 may be used.

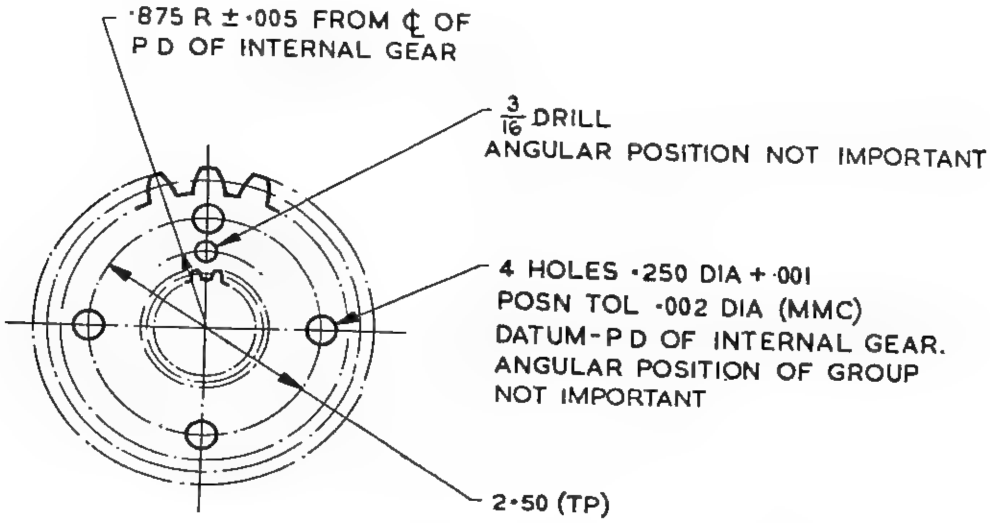


Fig. 114. Notes used where angular relationship may vary freely

20. TOLERANCE NOTES FOR ROUNDNESS

a. It is rarely necessary to specify tolerances for roundness because the errors of roundness present in cylindrical surfaces are normally within the limits required for assembly and functioning.

However, in certain cases, the permissible errors of roundness are, for functional reasons, so small that the necessary accuracy cannot be guaranteed automatically by the normal machining processes. In these cases it is necessary to specify tolerances for roundness.

b. Where it is necessary to specify the maximum ovality that is permissible in a cylindrical feature, a note as shown in Fig. 115 should be used.

Ovality, for the purpose of this note, is the maximum difference in diametral measurements on any cross section of the cylinder.

c. However, diametral measurements cannot, by themselves, prove that a feature is round. Figures such as those shown in Fig. 116 obviously suffer from errors of roundness even though their ' diametral ' measurements are constant.

d. Theoretically, errors of roundness or cylindricity (i.e. roundness, straightness and parallelism) could be toleranced by establishing an annular tolerance zone within which the finished surface must lie (see Fig. 117). However, such a requirement, while being easy to specify, is most difficult to check in practice because of the difficulty of establishing the perfect cylinder with which to compare the finished surface, and also because of the infinite variety of errors of form that may occur.

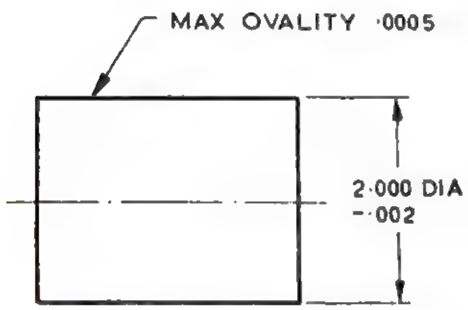


Fig. 115. Note defining maximum ovality

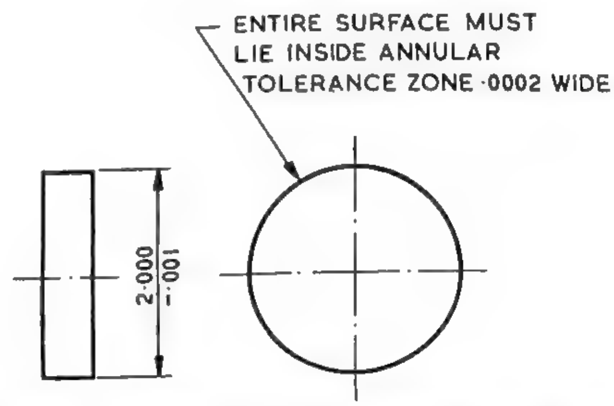


Fig. 117. Tolerancing errors of roundness by annular tolerance zone. *Not recommended* for general application. See Clause 20 *d*.

(*a*) (*b*)

Fig. 116. Lobed figures with constant ' diametral ' dimensions

e. In practice, if rotation between centres is not practicable, errors of roundness and cylindricity can be detected by using a vee block with an indicator as in Fig. 118 (*a*) and (*b*), or by using comparators which measure across three points as illustrated in Fig. 118 (*c*) and (*d*).

The readings obtained by these methods cannot be related to a tolerance specified as shown in Fig. 117 without an involved mathematical analysis for each individual part being checked.

CLAUSE 20 (*cont.*)

It is therefore recommended that, where it is necessary to specify a tolerance for roundness or cylindricity, the tolerance should be expressed as the full indicator movement permissible when the work is checked using a particular angle of vee block or arrangement of fixed contacts.

Fig. 119 shows typical tolerance notes which indicate the tolerance and make reference to the angle of the vee block or the disposition of the fixed contacts on which the specified tolerance is based.*

* The ideal angle of V block depends upon the number of lobes which are present. For a three lobed figure as shown in Fig. 116 (*a*) the ideal angle of V block is 60°. For a five lobed figure (Fig. 116 (*b*)) it is 108° and for one containing seven lobes 129°. It has been found in practice that a V block having an included angle of 120° gives a reasonable comparative reading in most cases and it is therefore recommended that this angle of V or an equivalent disposition of fixed contacts be used whenever practicable.

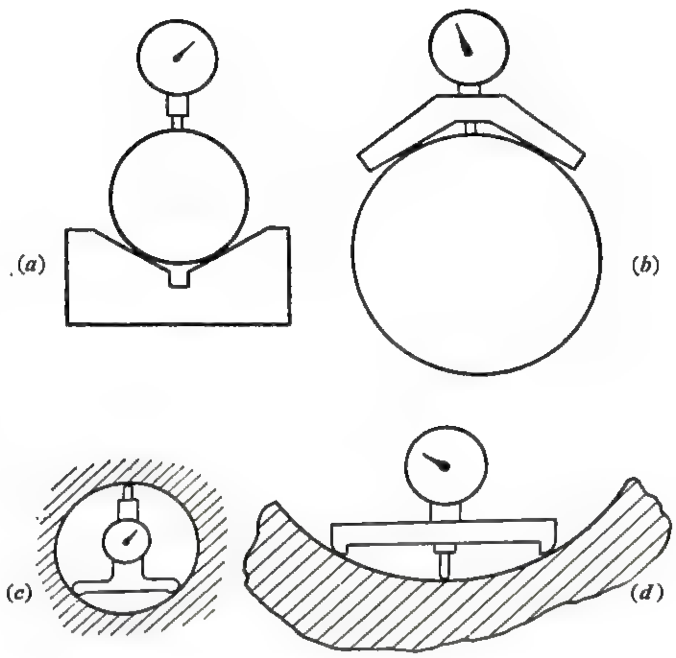


Fig. 118. Practical methods of checking errors of roundness

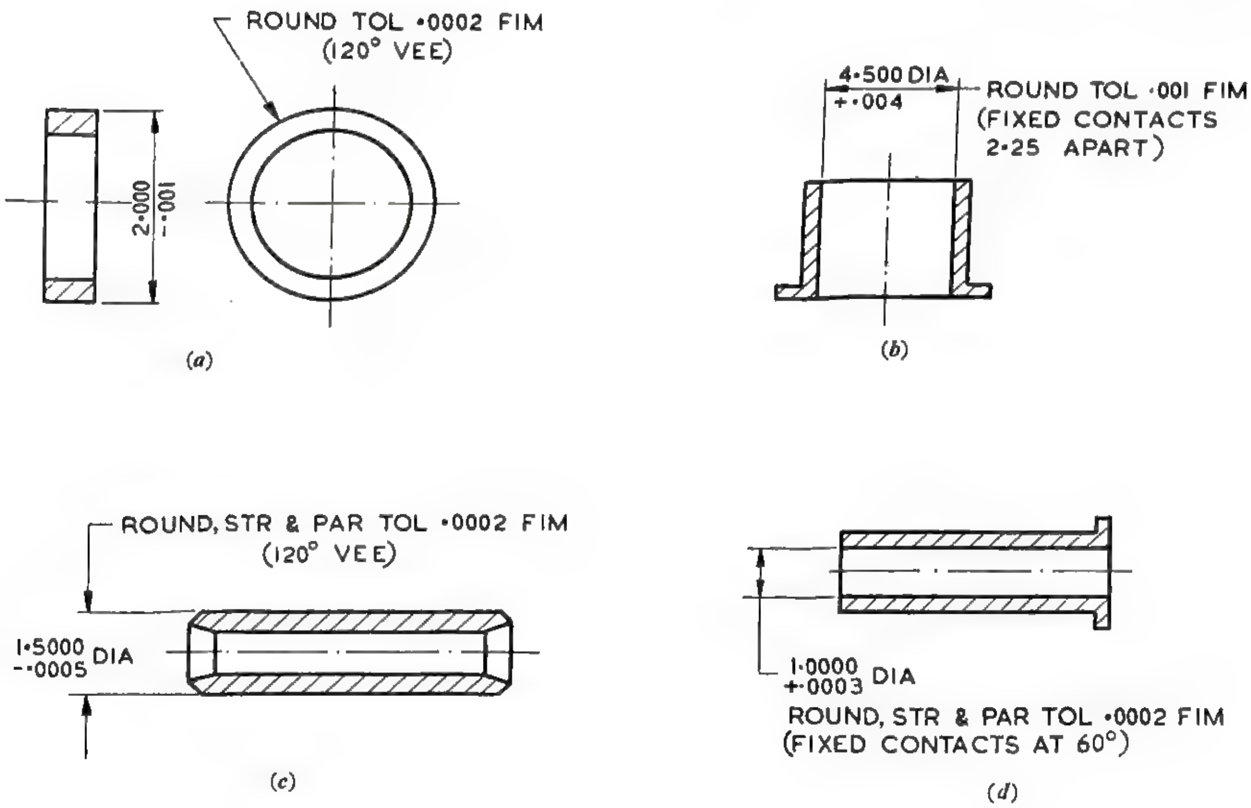
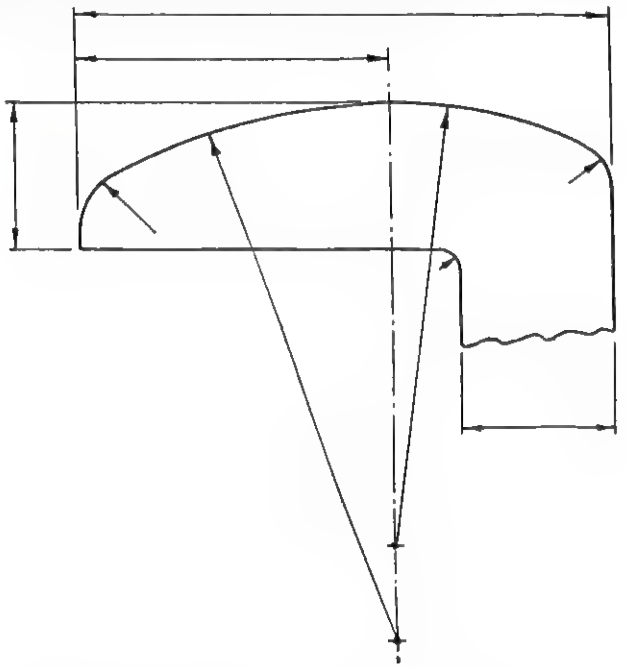


Fig. 119. Recommended tolerance notes for roundness and cylindricity

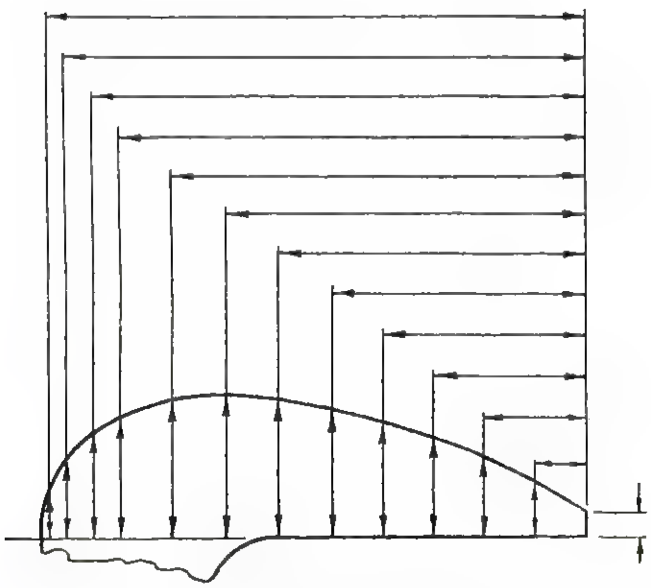
21. PROFILES AND CURVED SURFACES

a. Dimensioning. A curved line composed of circular arcs should preferably be dimensioned by radii as in Fig. 120 (*a*). Ordinates, as in Fig. 120 (*b*), should only be used if the preferred method is impracticable.

Where the ordinate method is used, the ordinates should be close enough to reduce possible deviations of the curve to a reasonable amount. The co-ordinates may be rectangular or polar and, where convenient, may be given in tabular form. When dimensioning cam profiles, it is often found convenient to give the dimensions in association with a replica of the follower (see Fig. 126).



(*a*) By means of radii

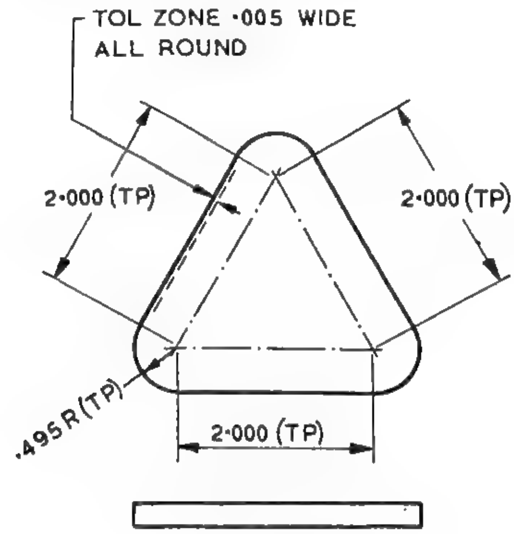


(*b*) By means of ordinates

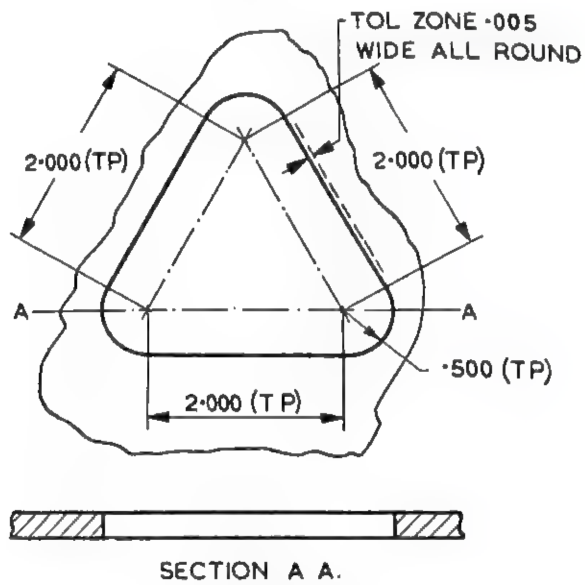
Fig. 120. Dimensioning of curved lines

b. Tolerancing. There are two methods of tolerancing the form of a profile. The first of these methods shows on the drawing the tolerance zone within which the finished profile is to lie. In the second method the ordinates which define the profile are directly toleranced.*

(i) Examples of the use of a tolerance zone are shown in Figs. 121 and 122. In Fig. 121 the tolerance zone



(*a*) External profile.



(*b*) Internal profile.

Fig. 121. Profiles with unilateral tolerance zones

* One important difference between the two methods is that the profile tolerance zone method provides a uniform metal tolerance normal to the profile; whereas, in the toleranced ordinate method, the metal tolerance, normal to the surface, will vary with the shape of the profile.

CLAUSE 21 (*cont.*)

is unilateral while that in Fig. 122 is bilateral. Note that the dimensions which define the design profile are labelled 'TP' (i.e., true profile). This is to distinguish them from other dimensions which may be limited by a general tolerance note. Where a general tolerance note is not used and all dimensions other than datum and auxiliary dimensions, and those defining true profiles (or positions), are individually toleranced, these distinguishing letters (TP) may be omitted. In such cases a general note may be used to indicate that all untoleranced dimensions define true profiles or positions.

Where a profile needs to be correctly related to other features, the other features may be used as datum features for the profile as in Fig. 123; alternatively, the maximum metal profile may be used as the datum for the related features as in Fig. 124.

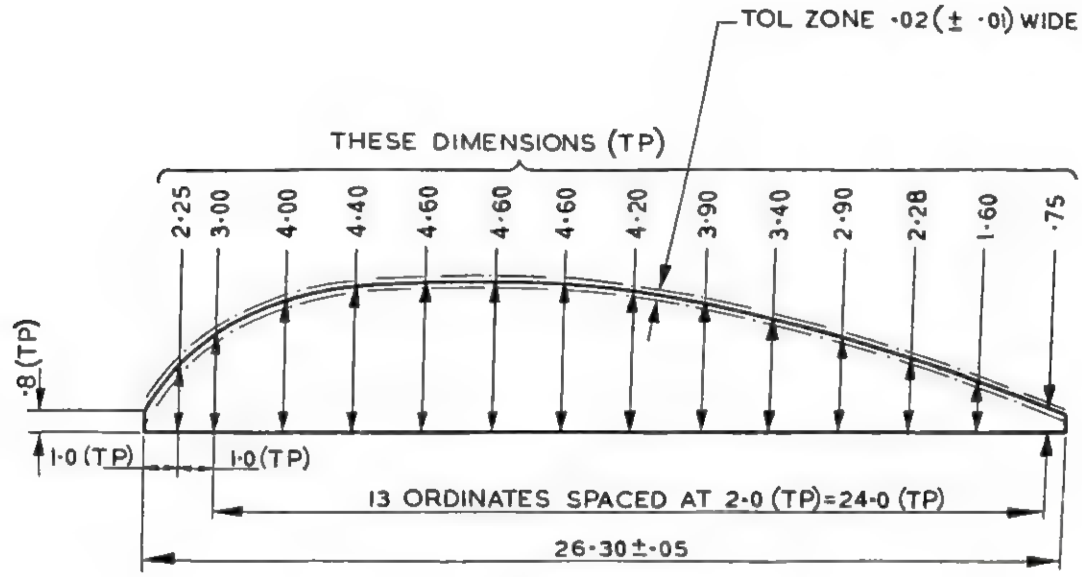


Fig. 122. Profile with bilateral tolerance zone

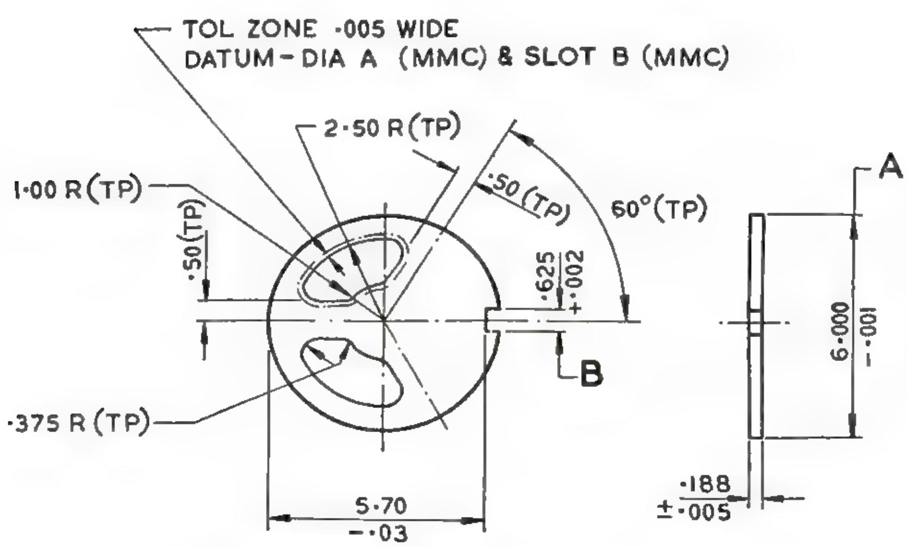


Fig. 123. Profiles toleranced with relation to datum features

CLAUSE 21 (*cont.*)

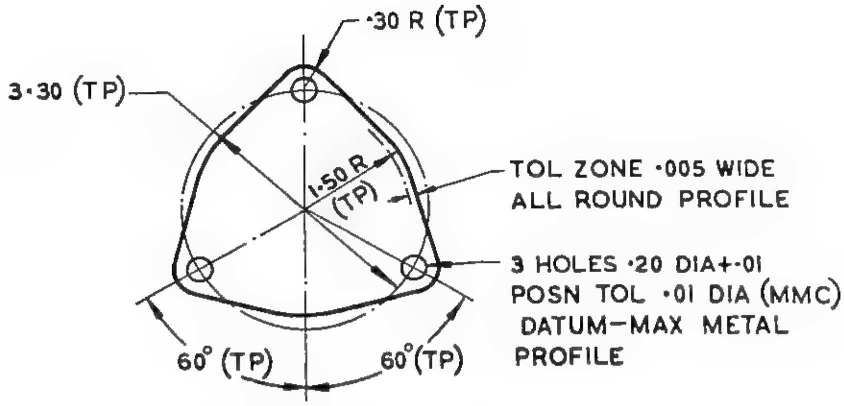


Fig. 124. Profile used as datum for position

(ii) Where the ordinates which define the form are directly toleranced, the dimensions which locate the ordinates should be labelled ' DATUM ' as shown in Figs. 125 and 126.

Wherever there is any possibility of misunderstanding, the datum features for the location and measurement of the ordinates should be indicated by a note as in Fig. 126.

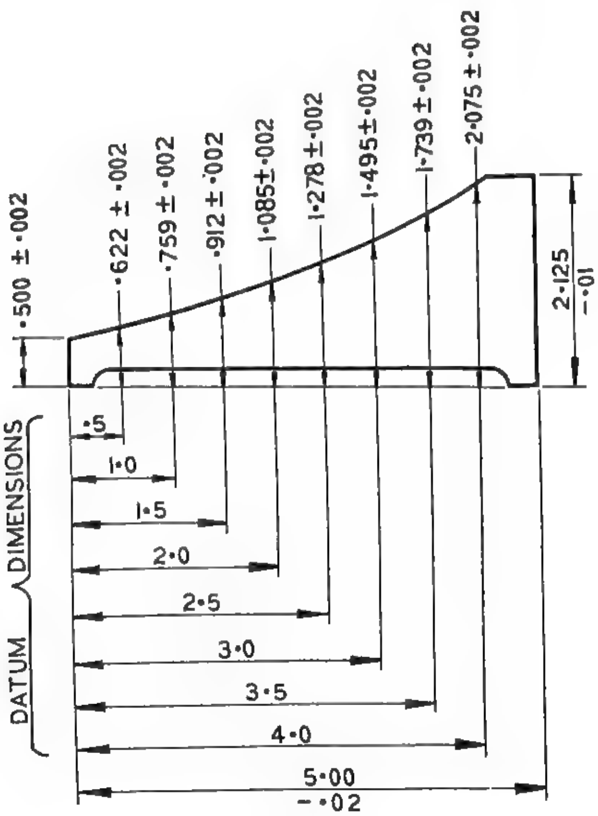
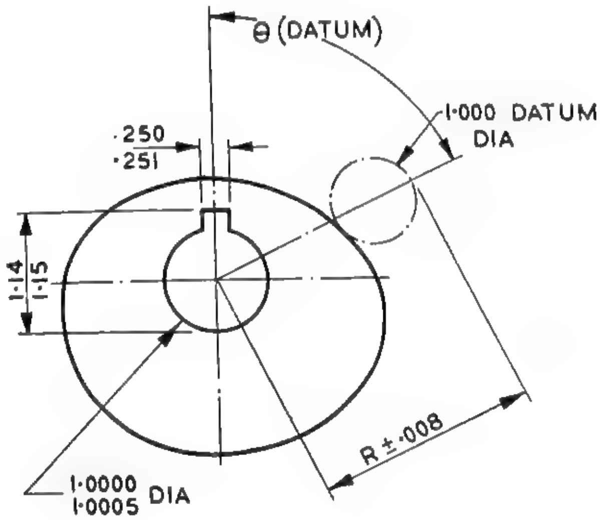


Fig. 125. Example of toleranced ordinates



DATUM FOR R & θ, BORE AND KEYWAY (MMC)

θ°	0	20	40	60	80	100	
R	2·000	2·067	2·250	2·500	2·750	2·933	

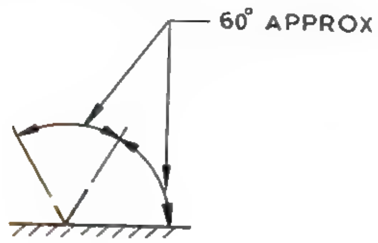
θ°	120-210	230	260	280	300	320	340
R	3·000	2·957	2·750	2·552	2·345	2·165	2·043

Fig. 126. Cam profile dimensioned by toleranced polar ordinates given over a replica of a follower

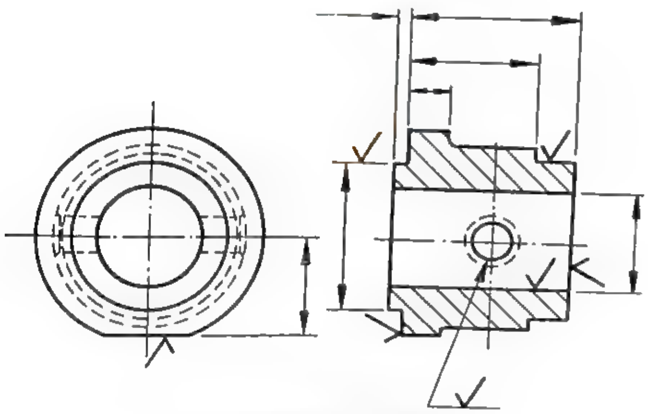
22. MACHINING AND ROUGHNESS SYMBOLS

a. *Machining symbol.*

(i) Where it is necessary to indicate that a surface is to be machined, without defining either the grade of roughness or the process to be used, the symbol shown in Fig. 127 (*a*) should be applied as illustrated in Fig. 127 (*b*). This symbol implies that a machining allowance shall be provided.



(*a*) Symbol.



(*b*) Application of symbols.

Fig. 127. Machining symbols

(ii) Where all the surfaces are to be machined a general note such as the following may be used :

√ ALL OVER

(iii) Particular machining processes should not normally be specified on other than process drawings except in the special circumstances mentioned in Sub-clause *c* below.

(iv) Machining symbols, like dimensions, should not normally be duplicated and it is desirable that they be placed on the same view as the dimensions which give the size or location of the surfaces concerned.

(v) Machining symbols may sometimes be omitted where it is known that the surface concerned can only be obtained by machining and where the finish produced by normal processes is acceptable.

(vi) Where it is necessary to limit the roughness of an unmachined surface, this should be done by a note as in Fig. 128.

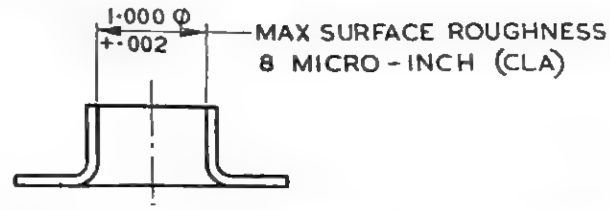
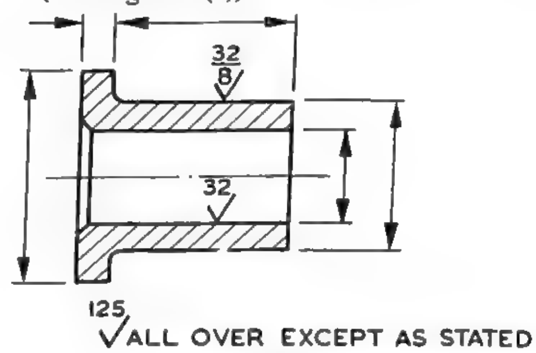


Fig. 128. Limiting the roughness of an unmachined surface

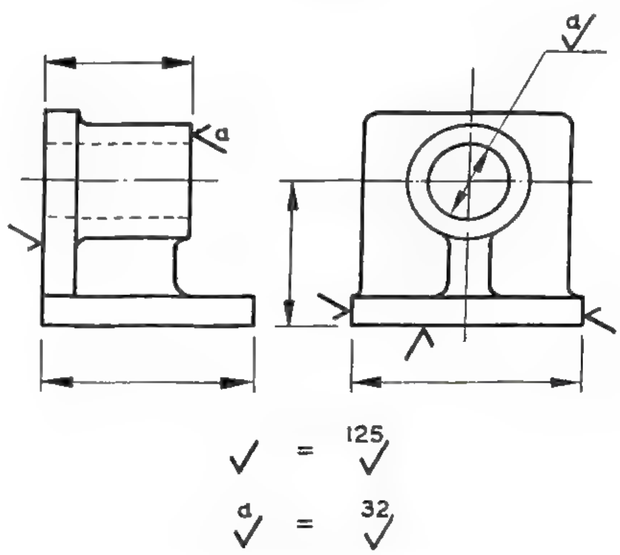
b. *Roughness numbers*.*

(i) Where it is necessary to specify the maximum roughness that can be permitted, the appropriate roughness number should be placed inside the symbol as shown in Fig. 129 (*a*) and (*b*). If both maximum and minimum values have to be specified, both the roughness numbers should be shown as in Fig. 129 (*a*).

As an alternative to placing the roughness number within the machining symbol, a letter may be used provided the meaning of the letter is clearly defined. (See Fig. 129 (*b*)).



(*a*)



(*b*)

Fig. 129. Application of roughness numbers

* For a full account of the measurement of surface roughness see B.S. 1134, ' The assessment of surface texture '.

CLAUSE 22 (*cont.*)

(ii) The roughness numbers represent the average departure of the surface from perfection over a prescribed 'sampling length' (normally 0·03 in.) and are expressed in micro-inches*. The sampling length need only be specified in those special circumstances where it must be other than 0·03 in. (see Sub-clause *c* (iii) below). The measurements are normally made along a line running at right angles to the lay or general direction of the toolmarks or scratches in the surface. The roughness of surfaces may be measured by special instruments or checked by comparison with specimen blocks.

(iii) The British Standard index numbers of surface roughness are as follows :

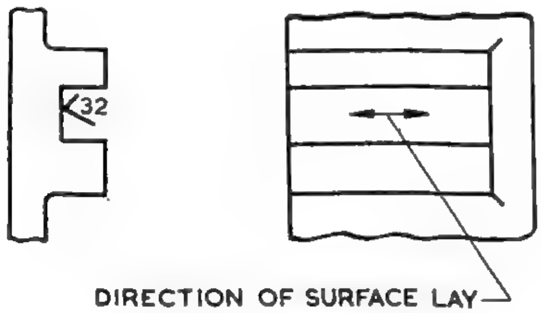
1, 2, 4, 8, 16, 32, 63, 125, 250, 500, 1000.

The choice of index numbers should be restricted to this list whenever possible.

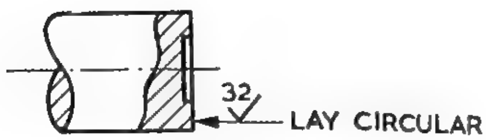
(iv) General notes may be used as in Fig. 129 (*a*) to specify roughness values where most of the surfaces have the same limiting value.

c. Special requirements.

(i) *Lay.* As the direction of the toolmarks or scratches on a surface is usually unimportant, it is rarely necessary to specify it on a drawing. Sometimes, however, it may be necessary for functional reasons to specify the direction of lay, which may be indicated on drawings by a note or other suitable means as in Fig. 130.



(*a*)

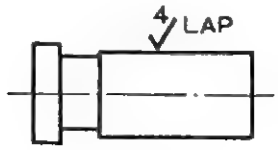


(*b*)

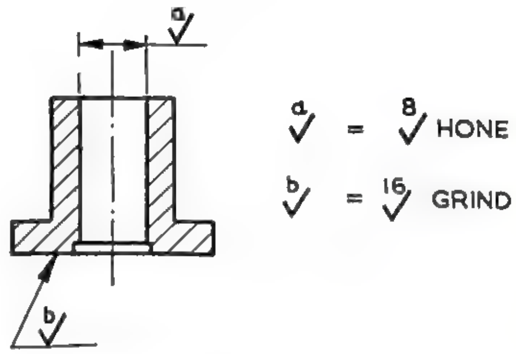
Fig. 130. Indication of direction of surface lay

* 1 micro-inch = 1 millionth part (0·000 001) of an inch.

(ii) *Machining processes.* Where the required characteristics of a surface can be produced only by one particular process, such as lapping, honing, scraping, etc., the appropriate process should be specified, as in Fig. 131.



(*a*)



(*b*)

Fig. 131. Indication of finishing process where essential for functioning

(iii) *Special sampling lengths.* When the sampling length is to be other than 0·03 in., it should be indicated by a note such as is shown in Fig. 132.

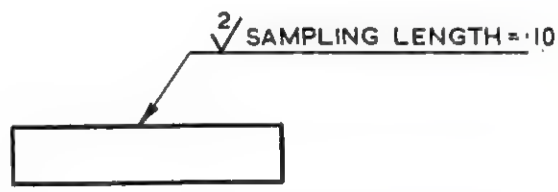


Fig. 132. Indication of special sampling length

d. Surfaces treated by plating, chemicals, etc. Where surfaces are to be given a final treatment such as plating or chemical processing, any roughness numbers specified should apply after such treatment. Any departure from the above principle should be limited to special cases and should be the subject of an explanatory note.

23. TYPICAL EXAMPLES OF ENGINEERING DRAWINGS

The drawings which follow have been prepared to illustrate some of the principles recommended in this standard. They are typical only and are not intended to show the sole manner in which each component should be treated. In fact it will be noted that two of the components have been treated in two different ways, one drawing of each component having been more fully toleranced than the other.

In order to economise in draughting time, the degree to which tolerancing is carried should be determined by consideration of such factors as the following:

(*a*) The nature or type of production, e.g. experimental, small quantity or mass production.

(*b*) The degree of interchangeability required.

(*c*) The nature and quality of the manufacturing equipment, where this is known.

(*d*) The experience and knowledge of the labour to be used in production and inspection, where this is known.

(*e*) Knowledge of any established shop or company standards of accuracy.

Plates 1, 2, 4 and 6.

These drawings are typical of machine parts to be produced under either of the following conditions :—

(*a*) Where the work may be destined for subcontracting and hence the circumstances of production are uncertain.

(*b*) Where adequate information, in regard to both the dimensional and geometrical characteristics of the finished product, is necessary to enable planning, jig and tool design, and inspection to be carried out on a sound basis.

Plates 3 and 5.

These drawings, which are less extensively toleranced than those in plates 1, 2, 4 and 6, are typical of machine parts to be produced under any of the following conditions :

(*a*) For experimental work or individually-built assemblies.

(*b*) Where the components shown are typical of the usual run of work in a factory, and the production and inspection staffs have a sound knowledge of the design requirements.

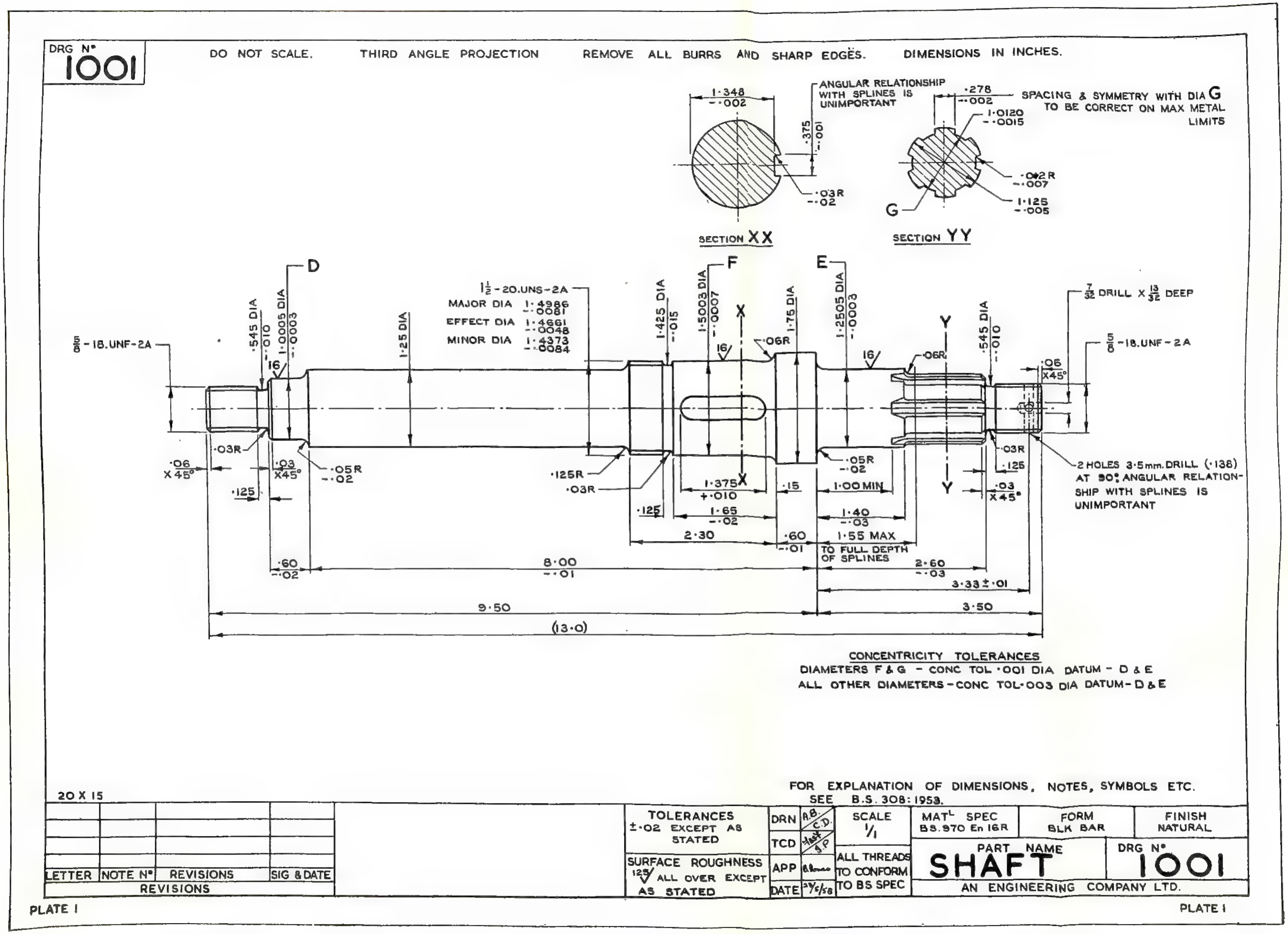
(*c*) Where there is in existence, in a particular factory, an established system of tolerances covering geometrical and other requirements.

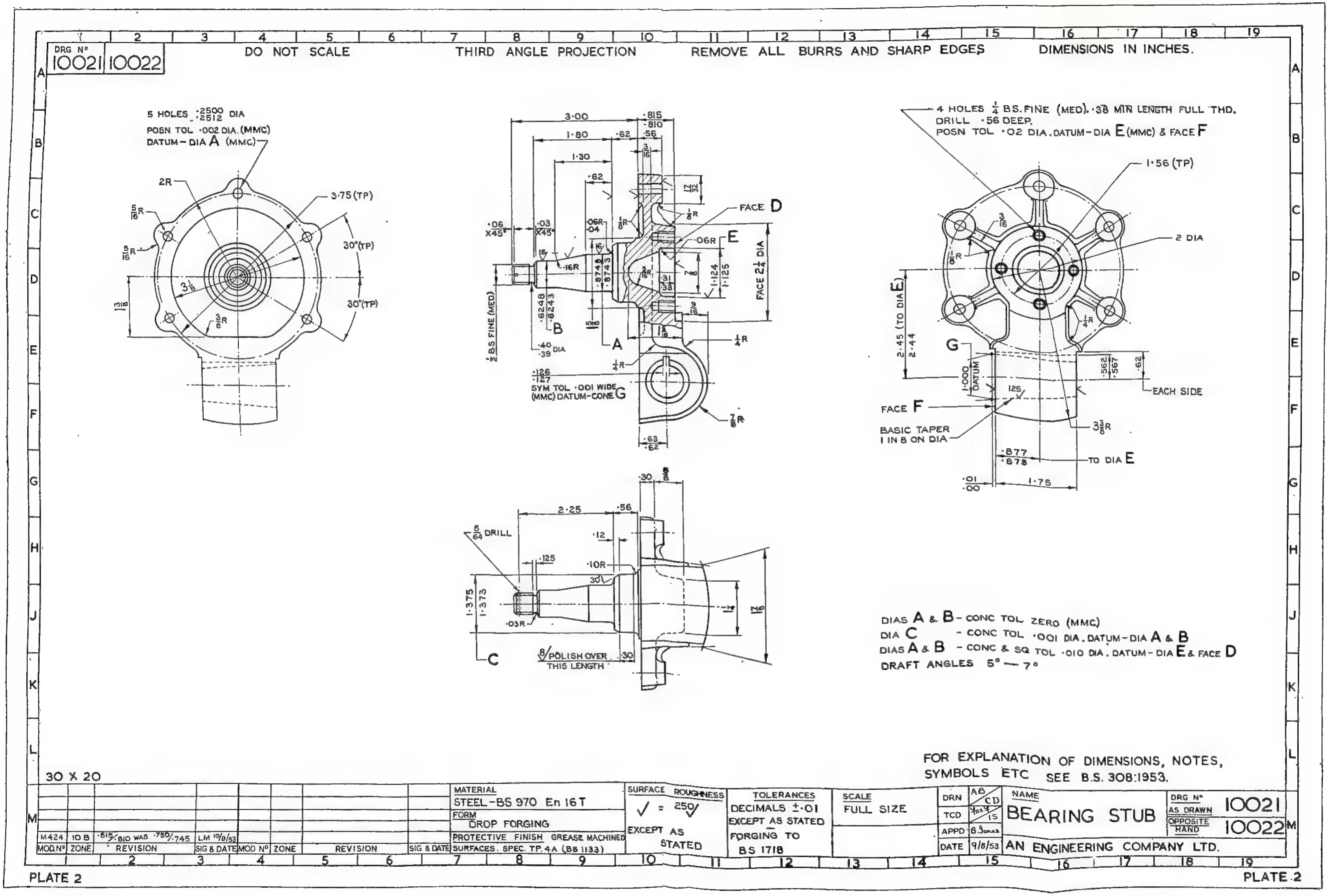
Plate 7.

This is a typical gauge drawing suitable for subcontracting. It will be noted that the principles laid down in this standard are also applicable to drawings of this character : they serve, in fact, for drawings of any engineering product.

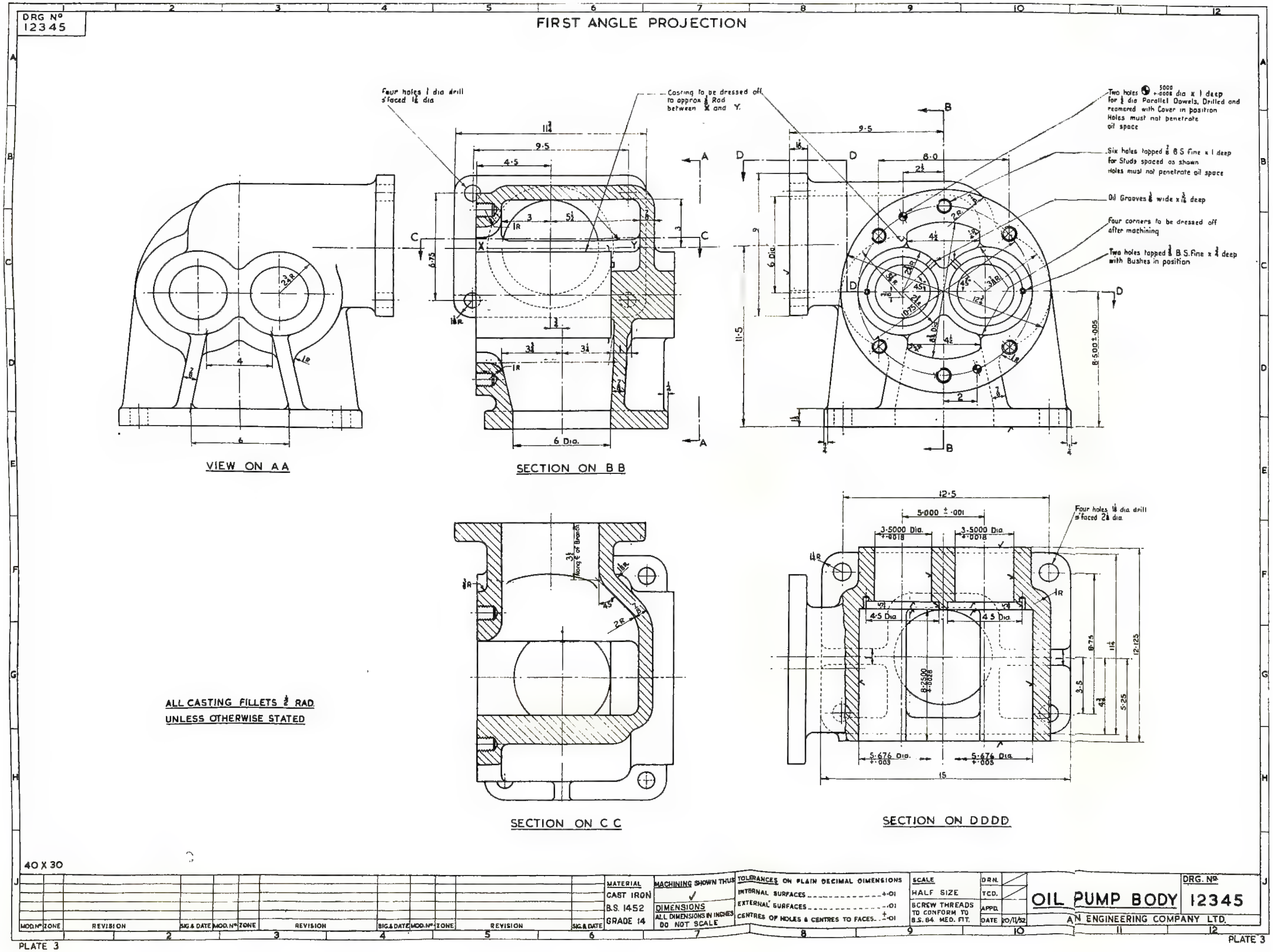
General Note. It will be appreciated that there can be no hard and fast rule for determining the extent of tolerancing ; each case is conditioned by the factors outlined above according to the particular circumstances. Satisfactory products, coupled with overall economy in design, production and inspection, must be the final objective.

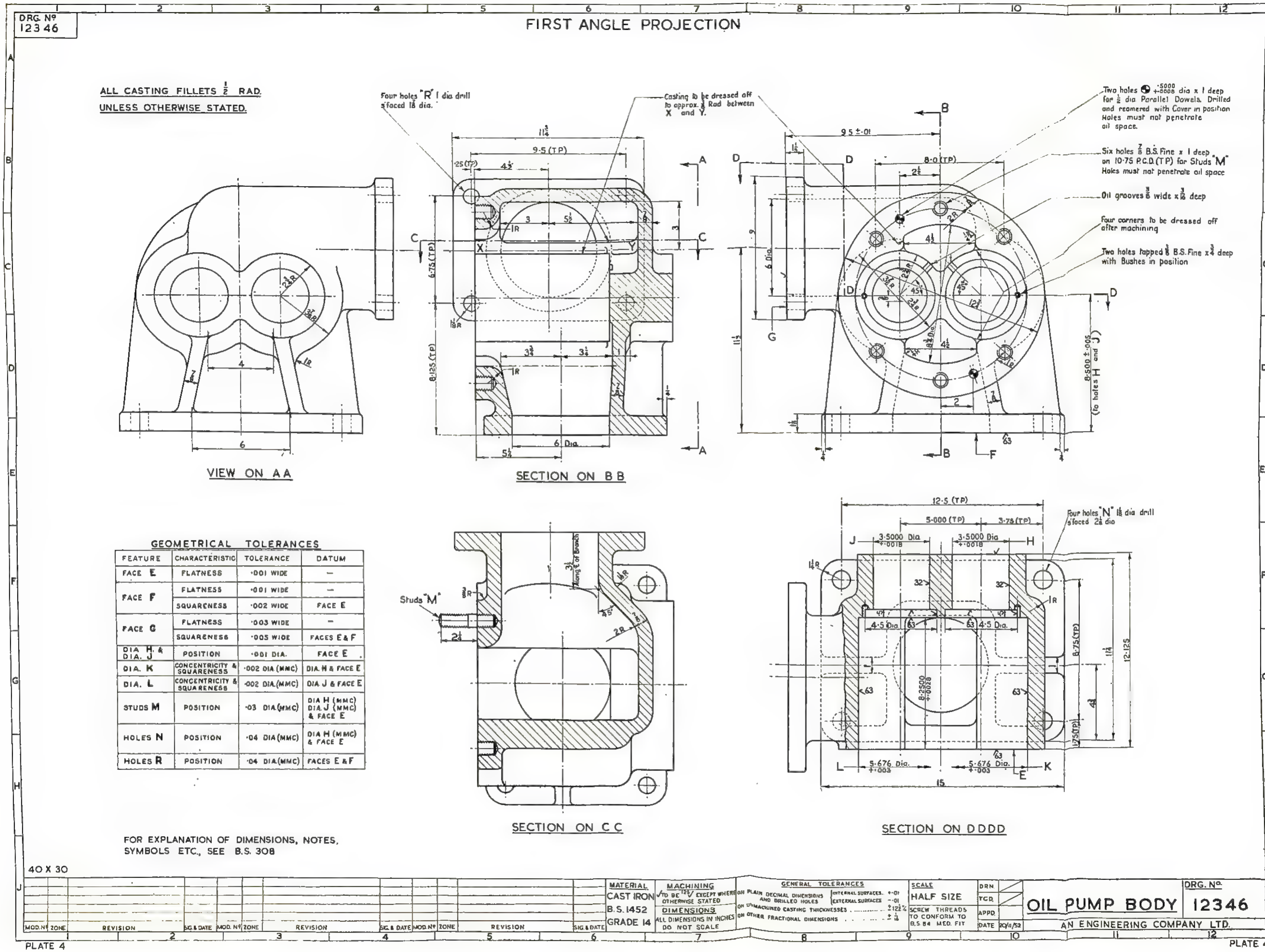
u\


DRG N° 1001
DO NOT SCALE. THIRD ANGLE PROJECTION REMOVE ALL BURRS AND SHARP EDGES. DIMENSIONS IN INCHES.
ANGULAR RELATIONSHIP WITH SPLINES IS UNIMPORTANT
1·348 −·002
·375 −·001
·03R −·02
SECTION XX
·278 −·002
1·0120 −·0015
SPACING & SYMMETRY WITH DIA G TO BE CORRECT ON MAX METAL LIMITS
·012 R −·007
1·125 −·005
G
SECTION YY
D
F
E
X
Y
1½ − 20.UNS − 2A
MAJOR DIA 1·4986 −·0081
EFFECT DIA 1·4661 −·0048
MINOR DIA 1·4373 −·0084
⅝ − 18.UNF − 2A
·545 DIA −·010
1·0005 DIA −·0003
1·25 DIA
1·425 DIA −·015
1·5003 DIA −·0007
1·75 DIA
1·2505 DIA −·0003
·545 DIA −·010
16
·06R
7/32 DRILL X 13/32 DEEP
⅝ − 18.UNF − 2A
·06 X45°
·03R
·06 X 45°
·03 X45°
·05R −·02
·125
·125R
·03R
·125
1·375 +·010
1·65 −·02
2·30
·15
·60 −·01
1·00 MIN
1·40 −·03
1·55 MAX TO FULL DEPTH OF SPLINES
·05R −·02
·03R
·125
·03 X45°
2 HOLES 3·5mm. DRILL (·138) AT 90°. ANGULAR RELATIONSHIP WITH SPLINES IS UNIMPORTANT
·60 −·02
8·00 −·01
2·60 −·03
3·33 ± ·01
9·50
3·50
(13·0)
CONCENTRICITY TOLERANCES
DIAMETERS F & G − CONC TOL ·001 DIA DATUM − D & E
ALL OTHER DIAMETERS − CONC TOL ·003 DIA DATUM − D & E
FOR EXPLANATION OF DIMENSIONS, NOTES, SYMBOLS ETC. SEE B.S. 308: 1953.
20 X 15
LETTER NOTE N° REVISIONS SIG & DATE
REVISIONS
TOLERANCES ±·02 EXCEPT AS STATED
SURFACE ROUGHNESS 125 ALL OVER EXCEPT AS STATED
DRN
TCD
APP
DATE
SCALE 1/1
ALL THREADS TO CONFORM TO BS SPEC
MATL SPEC BS.970 En 16R
FORM BLK BAR
FINISH NATURAL
PART NAME SHAFT
DRG N° 1001
AN ENGINEERING COMPANY LTD.
PLATE I
PLATE I



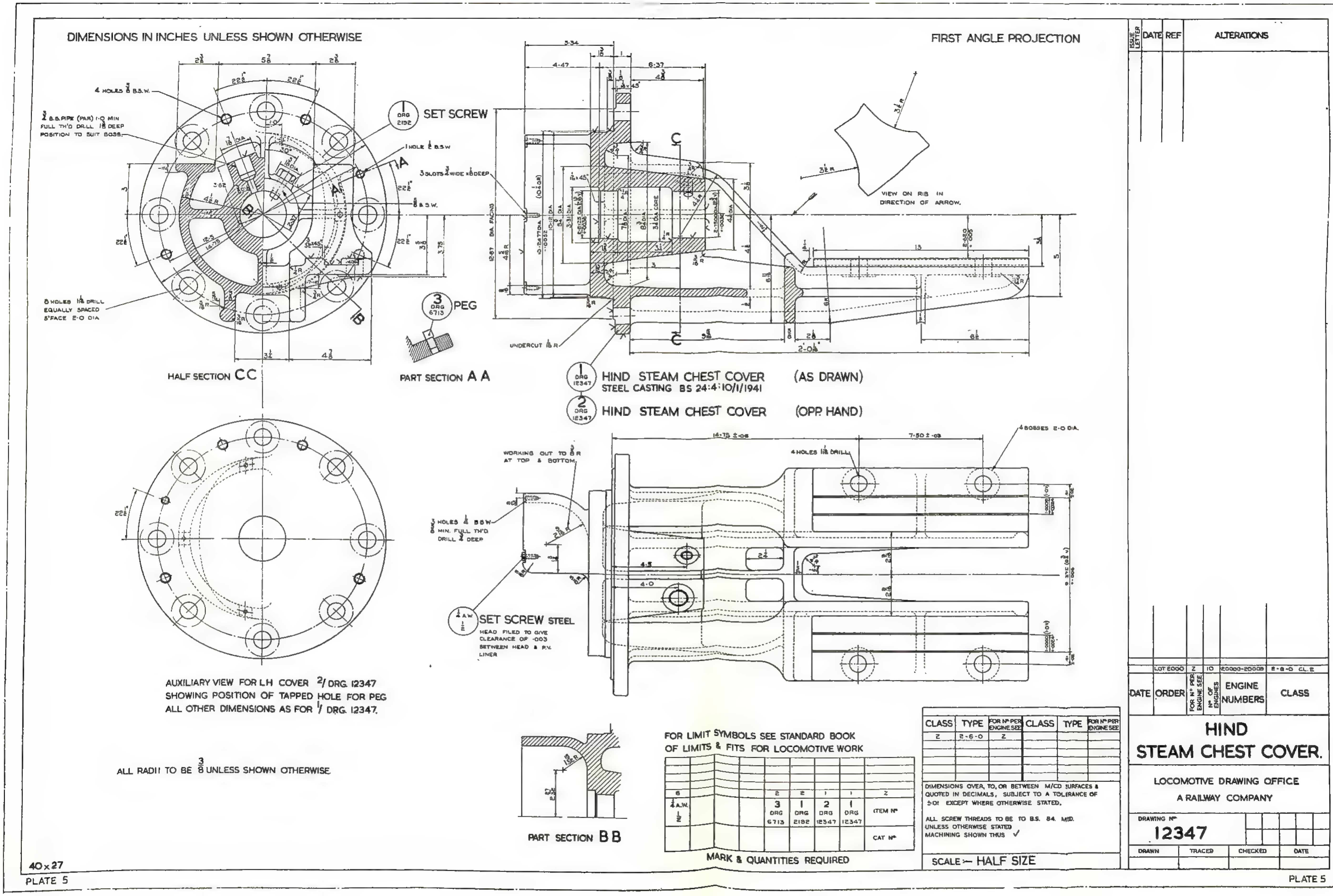
DRG No 10021 10022
DO NOT SCALE
THIRD ANGLE PROJECTION
REMOVE ALL BURRS AND SHARP EDGES
DIMENSIONS IN INCHES.
5 HOLES .2500/.2512 DIA
POSN TOL ·002 DIA. (MMC)
DATUM – DIA A (MMC)
4 HOLES 1/4 BS.FINE (MED). ·38 MIN LENGTH FULL THD.
DRILL ·56 DEEP.
POSN TOL ·02 DIA. DATUM – DIA E (MMC) & FACE F
FACE D
FACE 2 1/4 DIA
SYM TOL ·001 WIDE (MMC) DATUM – CONE G
BASIC TAPER 1 IN 8 ON DIA
EACH SIDE
TO DIA E
5/64 DRILL
POLISH OVER THIS LENGTH
DIAS A & B – CONC TOL ZERO (MMC)
DIA C – CONC TOL ·001 DIA. DATUM – DIA A & B
DIAS A & B – CONC & SQ TOL ·010 DIA. DATUM – DIA E & FACE D
DRAFT ANGLES 5° — 7°
FOR EXPLANATION OF DIMENSIONS, NOTES, SYMBOLS ETC SEE B.S. 308:1953.
30 X 20
M424 10 B ·815/·810 WAS ·750/·745 LM 10/8/53
MOD No ZONE REVISION SIG & DATE
MATERIAL STEEL – BS 970 En 16 T
FORM DROP FORGING
PROTECTIVE FINISH GREASE MACHINED SURFACES. SPEC. TP. 4A (BS 1133)
SURFACE ROUGHNESS √ = 250√ EXCEPT AS STATED
TOLERANCES DECIMALS ±·01 EXCEPT AS STATED FORGING TO BS 1718
SCALE FULL SIZE
DRN TCD APPD DATE 9/8/53
NAME BEARING STUB
DRG No AS DRAWN 10021 OPPOSITE HAND 10022
AN ENGINEERING COMPANY LTD.
PLATE 2


DRG Nº 12345
FIRST ANGLE PROJECTION
VIEW ON AA
SECTION ON BB
SECTION ON CC
SECTION ON DDDD
ALL CASTING FILLETS ⅜ RAD. UNLESS OTHERWISE STATED
40 X 30
MATERIAL CAST IRON B.S. 1452 GRADE 14
MACHINING SHOWN THUS ✓
DIMENSIONS ALL DIMENSIONS IN INCHES DO NOT SCALE
TOLERANCES ON PLAIN DECIMAL DIMENSIONS
SCALE HALF SIZE
SCREW THREADS TO CONFORM TO B.S. 84 MED. FIT.
OIL PUMP BODY
DRG. Nº 12345
AN ENGINEERING COMPANY LTD.
PLATE 3

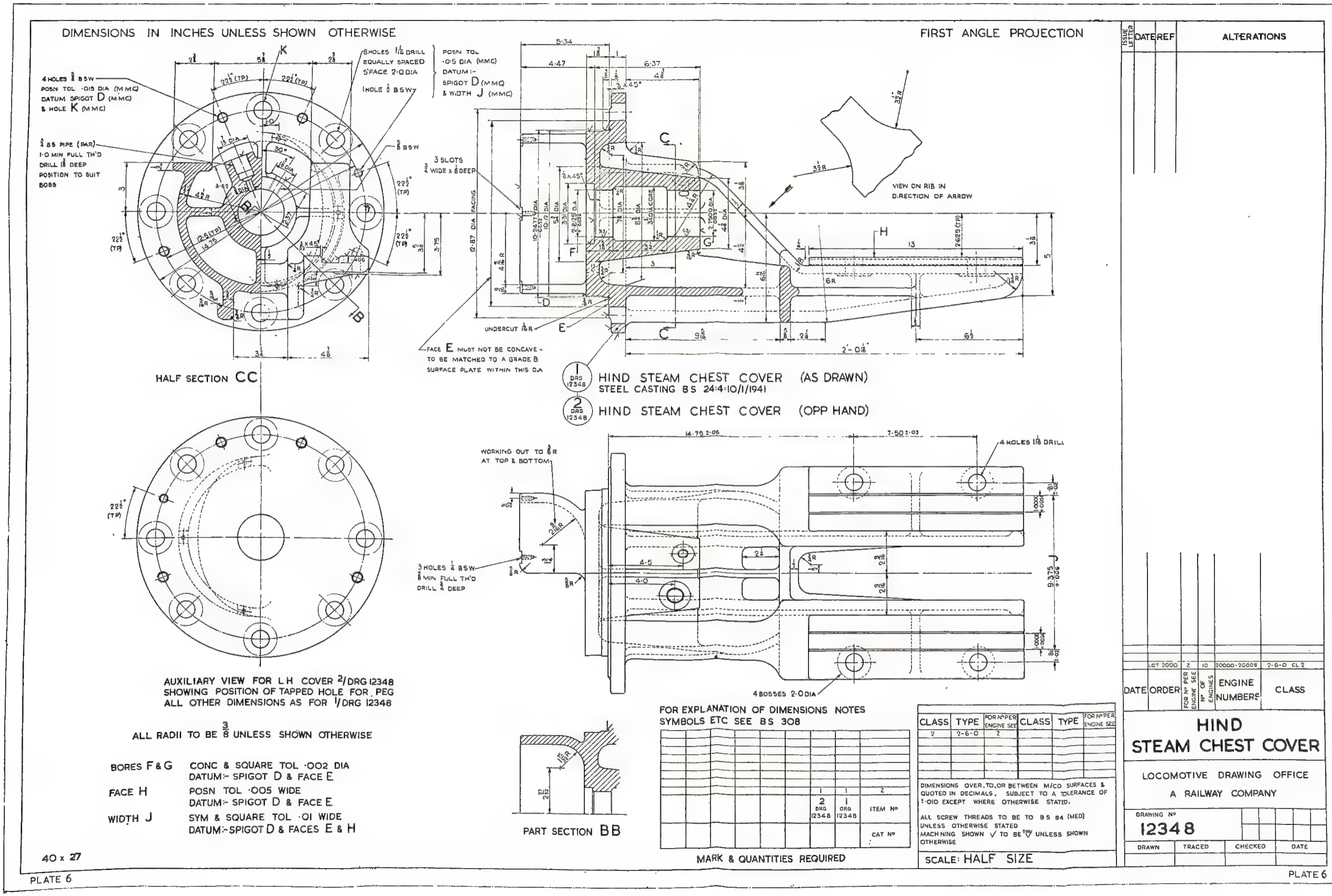


GEOMETRICAL TOLERANCES

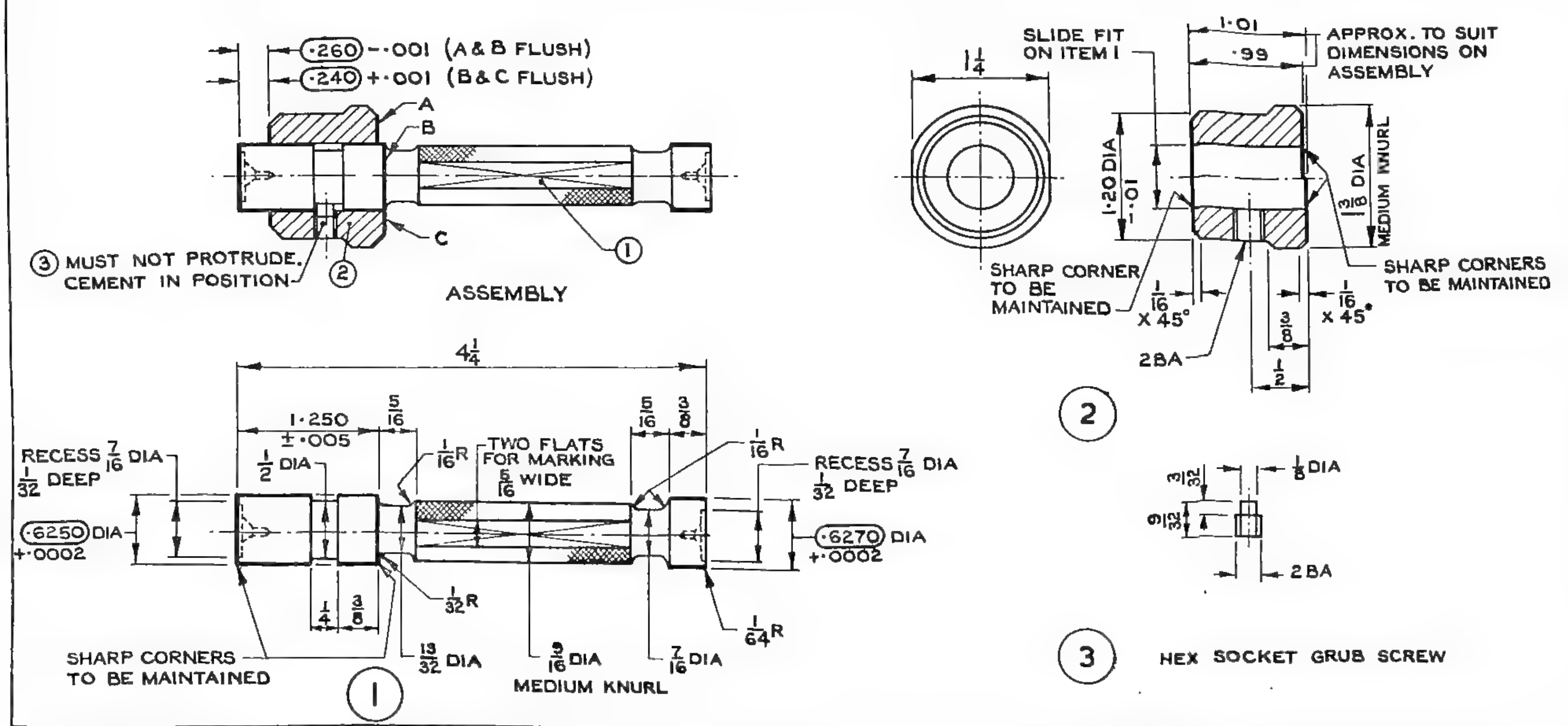
FEATURE	CHARACTERISTIC	TOLERANCE	DATUM
FACE E	FLATNESS	·001 WIDE	—
FACE F	FLATNESS	·001 WIDE	—
	SQUARENESS	·002 WIDE	FACE E
FACE G	FLATNESS	·003 WIDE	—
	SQUARENESS	·005 WIDE	FACES E & F
DIA. H. & DIA. J	POSITION	·001 DIA.	FACE E
DIA. K	CONCENTRICITY & SQUARENESS	·002 DIA. (MMC)	DIA. H & FACE E
DIA. L	CONCENTRICITY & SQUARENESS	·002 DIA. (MMC)	DIA. J & FACE E
STUDS M	POSITION	·03 DIA. (MMC)	DIA H (MMC) DIA. J (MMC) & FACE E
HOLES N	POSITION	·04 DIA (MMC)	DIA H (MMC) & FACE E
HOLES R	POSITION	·04 DIA. (MMC)	FACES E & F



DIMENSIONS IN INCHES UNLESS SHOWN OTHERWISE
FIRST ANGLE PROJECTION
SET SCREW
PEG
HALF SECTION CC
PART SECTION AA
HIND STEAM CHEST COVER (AS DRAWN)
STEEL CASTING BS 24:4:10/1/1941
HIND STEAM CHEST COVER (OPP. HAND)
VIEW ON RIB IN DIRECTION OF ARROW.
UNDERCUT
SET SCREW STEEL
HEAD FILED TO GIVE CLEARANCE OF ·003 BETWEEN HEAD & P.V. LINER
AUXILIARY VIEW FOR L.H. COVER 2/ DRG. 12347 SHOWING POSITION OF TAPPED HOLE FOR PEG ALL OTHER DIMENSIONS AS FOR 1/ DRG. 12347.
ALL RADII TO BE 3/8 UNLESS SHOWN OTHERWISE
PART SECTION BB
FOR LIMIT SYMBOLS SEE STANDARD BOOK OF LIMITS & FITS FOR LOCOMOTIVE WORK
MARK & QUANTITIES REQUIRED
ITEM Nº
CAT Nº
CLASS
TYPE
DIMENSIONS OVER, TO, OR BETWEEN M/CD SURFACES & QUOTED IN DECIMALS, SUBJECT TO A TOLERANCE OF ±·01 EXCEPT WHERE OTHERWISE STATED.
ALL SCREW THREADS TO BE TO B.S. 84. MED. UNLESS OTHERWISE STATED
MACHINING SHOWN THUS ✓
SCALE:— HALF SIZE
ISSUE LETTER
DATE
REF
ALTERATIONS
ORDER
ENGINE NUMBERS
HIND STEAM CHEST COVER.
LOCOMOTIVE DRAWING OFFICE
A RAILWAY COMPANY
DRAWING Nº 12347
DRAWN
TRACED
CHECKED
40 x 27
PLATE 5
PLATE 5


DIMENSIONS IN INCHES UNLESS SHOWN OTHERWISE
FIRST ANGLE PROJECTION
ISSUE LETTER
DATE
REF
ALTERATIONS
4 HOLES ⅜ BSW
POSN TOL ·015 DIA (MMC)
DATUM SPIGOT D (MMC)
& HOLE K (MMC)
8 HOLES 1 1/16 DRILL EQUALLY SPACED S'FACE 2·0 DIA
1 HOLE ½ BSW
POSN TOL ·015 DIA (MMC)
DATUM :- SPIGOT D (MMC) & WIDTH J (MMC)
⅜ BS PIPE (PAR) 1·0 MIN FULL TH'D DRILL 1⅛ DEEP POSITION TO SUIT BOSS
⅜ BSW
3 SLOTS WIDE x ⅛ DEEP
VIEW ON RIB IN DIRECTION OF ARROW
UNDERCUT
FACE E MUST NOT BE CONCAVE - TO BE MATCHED TO A GRADE B SURFACE PLATE WITHIN THIS DIA
HALF SECTION CC
1 DRG 12348 HIND STEAM CHEST COVER (AS DRAWN)
STEEL CASTING BS 24:4:10/1/1941
2 DRG 12348 HIND STEAM CHEST COVER (OPP HAND)
WORKING OUT TO ⅛R AT TOP & BOTTOM
4 HOLES 1⅛ DRILL
3 HOLES ¼ BSW ⅜ MIN FULL TH'D DRILL ¾ DEEP
4 BOSSES 2·0 DIA
AUXILIARY VIEW FOR LH COVER 2/DRG 12348
SHOWING POSITION OF TAPPED HOLE FOR PEG
ALL OTHER DIMENSIONS AS FOR 1/DRG 12348
ALL RADII TO BE ⅜ UNLESS SHOWN OTHERWISE
BORES F & G CONC & SQUARE TOL ·002 DIA
DATUM:- SPIGOT D & FACE E
FACE H POSN TOL ·005 WIDE
DATUM:- SPIGOT D & FACE E
WIDTH J SYM & SQUARE TOL ·01 WIDE
DATUM:- SPIGOT D & FACES E & H
PART SECTION BB
FOR EXPLANATION OF DIMENSIONS NOTES SYMBOLS ETC SEE BS 308
2 DRG 12348
1 DRG 12348
ITEM No
CAT No
MARK & QUANTITIES REQUIRED
CLASS
TYPE
FOR No PER ENGINE SET
2
2-6-0
DIMENSIONS OVER, TO, OR BETWEEN M/CD SURFACES & QUOTED IN DECIMALS, SUBJECT TO A TOLERANCE OF ±·010 EXCEPT WHERE OTHERWISE STATED.
ALL SCREW THREADS TO BE TO BS 84 (MED) UNLESS OTHERWISE STATED
MACHINING SHOWN √ TO BE UNLESS SHOWN OTHERWISE
SCALE: HALF SIZE
LOT 2000
20000-20009
2-6-0 CL 2
DATE
ORDER
FOR No PER ENGINE SET
No OF ENGINES
ENGINE NUMBERS
CLASS
HIND
STEAM CHEST COVER
LOCOMOTIVE DRAWING OFFICE
A RAILWAY COMPANY
DRAWING No
12348
DRAWN
TRACED
CHECKED
DATE
40 x 27
PLATE 6
PLATE 6

DO NOT SCALE THIRD ANGLE PROJECTION REMOVE BURRS AND SHARP CORNERS EXCEPT AS STATED



DIMENSIONS IN INCHES SCALE FULL SIZE
MATERIAL STEEL FRACTIONAL DIMENSIONS $\pm\frac{1}{64}$
DEGREE OF HARDNESS D.P.N. 650 TO 850

GAUGING DIMENSIONS ENCIRCLED
MACHINE ALL OVER
GAUGING SURFACES SHOWN BY THICK LINES TO BE HARDENED AND GROUND OR LAPPED. MAX ROUGHNESS 8 (BS 1134)
CENTRES TO BE GROUND OR LAPPED AND, WHERE SHOWN, LEFT IN

15 x 10

DATE	AMENDMENTS

GAUGE TO BE CLEARLY MARKED WITH: GAUGE No., SERIAL No., CONTRACTORS INITIALS, OR TRADE MARK, GO ·625, DEPTH ·240 – ·260, NOT GO ·627

GAUGE NUMBER
AZ 1634/2

DRN TCD CHD

PLATE 7. PLATE 7

ALPHABETICAL INDEX

TABLES

PLATES

Universal Decimal Classification 621.7 : 744

Printed in England by Waterlow & Sons Limited, 49 Parliament Street, Westminster, London, S.W.1.

B